ANILAM

6000M
CNC Control
TRAINING GUIDE

Navigation Instructions

Bookmarks

Training Guide
Help Menu
- Basic G-Codes
- Compensations
- Lines
- Arcs
- Blends & Chamfer
- Multiples
- Pockets
- Paths
- Drilling & Tapping
- G-Codes
- M-Codes

Program Management
Simple Program
- Create Program
- Edit Program
- G-Code Editor
- Sample Part
- Using Help
- Start Programming
- Draw Graghics
- Set X & Y offsets
- Set Tool Length
- Auto / Run Program

CAM Utility
DXF Converter
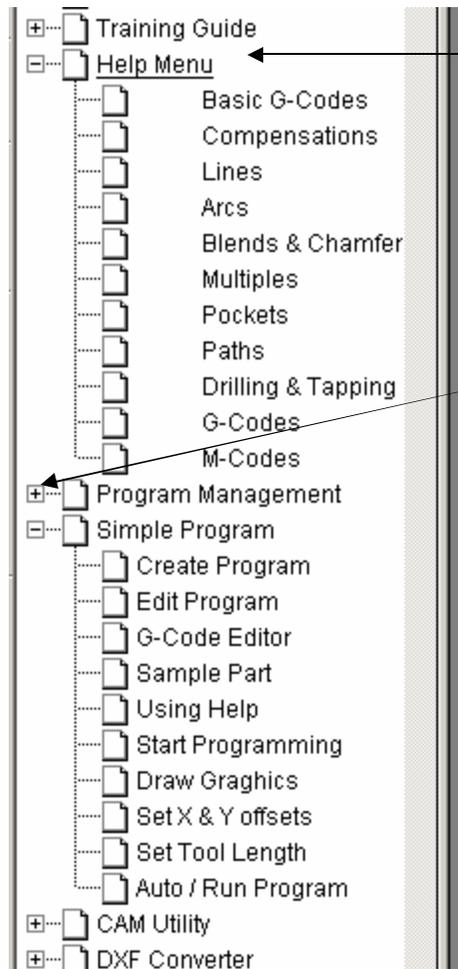
Follow the bookmarks at the left side of the page to navigate to desired topic.

Click plus and minus symbols to expand and compress menu display.

5000M CNC CONTROL TRAINING GUIDE

Turning the Control ON

```
 Version: 3.03A Off-Line      Anilam 5500M/MK CNC       Serial ID: 2263159

                          Manufactured By:

                             ANILAM Inc.
                          3300 Corporate Way
                         Miramar, Florida 33025
                           Tel: (954) 430-6600
                           Fax: (954) 450-6700
                            www.anilam.com

                       Copyright (c) 2000 by ANILAM Inc.
                             Miramar, FL, USA

                         Press F10 to continue.
```
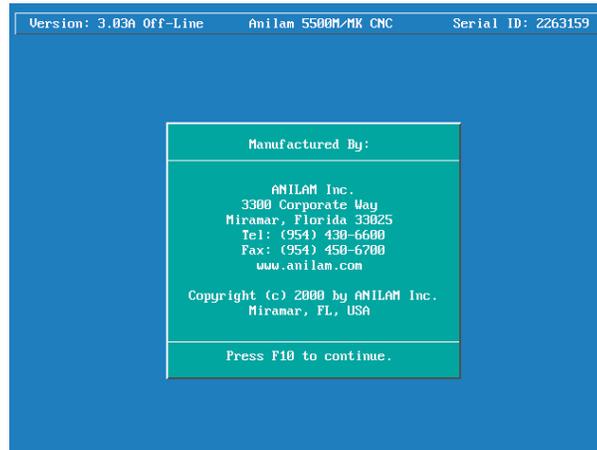
After the control has been turned ON press F10 | **F10** | to continue.

```
 Version: 3.03A Off-Line      Anilam 5500M/MK CNC       Serial ID: 2263159

                          Software Options

                       1. ►Control Software
                       2. Setup Utility
                       3. Motion Setup/Testing

           ↑      ↓    Select
```
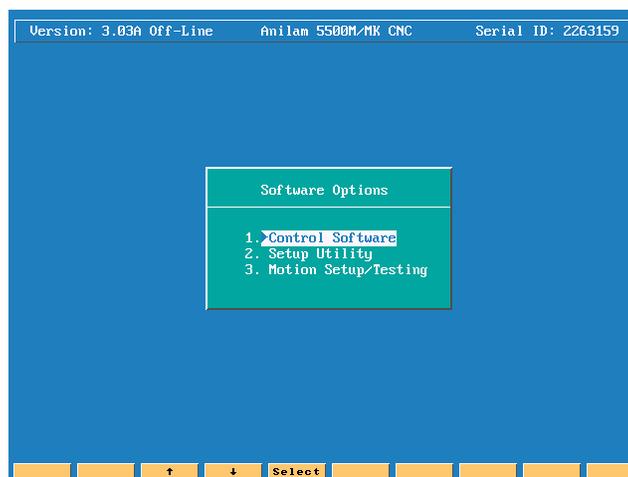
1. Control Software   This allows you enter CNC software
2. Setup Utility   Machine configuration
3. Motion Setup/Testing Used to setup drive

Then press ENTER   to select CNC mode.

1.

![ANILAM]

First CNC Screen

Control Manual Screen

```
              PROGRAM: 3AX-SP2.M      HALTED   MANUAL  IN-POSN

COMMAND:_                                                          SECTION #1

MESSAGE:

    MACHINE          PROGRAM           TARGET         DIST. TO GO

 X  +   0.0000    X  +   0.0000    X              X  +   0.0000
 Y  +   0.0000    Y  +   0.0000    Y              Y  +   0.0000
 Z  +   0.0000    Z  +   0.0000    Z              Z  +   0.0000    SECTION #2
 U  +   0.000     U  +   0.000     U              U  +   0.000
 W  +   0.000     W  +   0.000     W              W  +   0.000

TOOL: 0        RPM: 0      %: 100     LOOP: 0        FIXTURE: 0
 DIA: 0.0000   FEED: 0.0   %: 100    DWELL: 0.0
LENG: 0.0000                       OVERRIDE: FEED,RAPID

  G: G01 G17 G40 G70 G90             PARTS: 0                      SECTION #3
  M: M05 M09                         TIMER: 00:00:00 (00:00:00)

 Help  Program  Edit  Manual  S.Step  Auto  Delete  Insert  Tool  HandWl
```

2.

Section #1

Mode of operation

Machine position status

Program name

Machine status

```
                    PROGRAM:              HALTED    MANUAL   IN-POSN

COMMAND:_


MESSAGE:
```

Messages display in this area
Error and warnings

Command line for **MANUAL**(MDI) instructions

Section #2

```
        MACHINE              PROGRAM              TARGET            DIST. TO GO

    X   +    0.0000      X   +    0.0000      X                 X   +    0.0000
    Y   +    0.0000      Y   +    0.0000      Y                 Y   +    0.0000
    Z   +    0.0000      Z   +    0.0000      Z                 Z   +    0.0000
    U   +    0.000       U   +    0.000       U                 U   +    0.000
    W   +    0.000       W   +    0.000       W                 W   +    0.000
```

Position relative to machine HOME

Target or position to reach

Distance to go to reach target

Position relative to part ZERO

ANILAM

Section #3

Active tool #

Tool diameter

Tool length offset

Spindle speed

Feedrate

Percentage of
feed and RPM's

Number of loops

Dwell in seconds

Active fixture offset

```
TOOL: 0          RPM: 0        %: 100        LOOP: 0          FIXTURE: 0
 DIA: 0.0000     FEED: 0.0     %: 100       DWELL: 0.0
LENG: 0.0000                             OVERRIDE: FEED,RAPID

   G: G01 G17 G40 G70 G90               PARTS: 0
   M: M05 M09                           TIMER: 00:00:00 (00:00:00)
```

Active G codes

Active M functions

Override of feed and rapid

Parts counter and timer

4.

Softkey in Manual Mode

Access help menu

Edit program in memory

Single Step Mode

Delete from command line

Access tool page

| Help | Program | Edit | Manual | S.Step | Auto | Delete | Insert | Tool | HandWl |

Access list of programs

Manual Mode

Automatic Mode

Insert in command line

Activate handwheel

Note: When handwheel is active colors will be reversed in box as shown below.

F10

HandWl
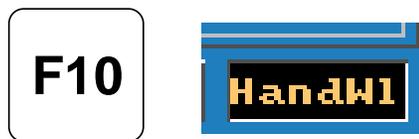
Softkey in Manual mode when shift key is pressed

| Message | | | | Teach | | Home | | | Exit |

Display last 8 messages

Teach Mode

Sends machine home

Exits back main manual screen
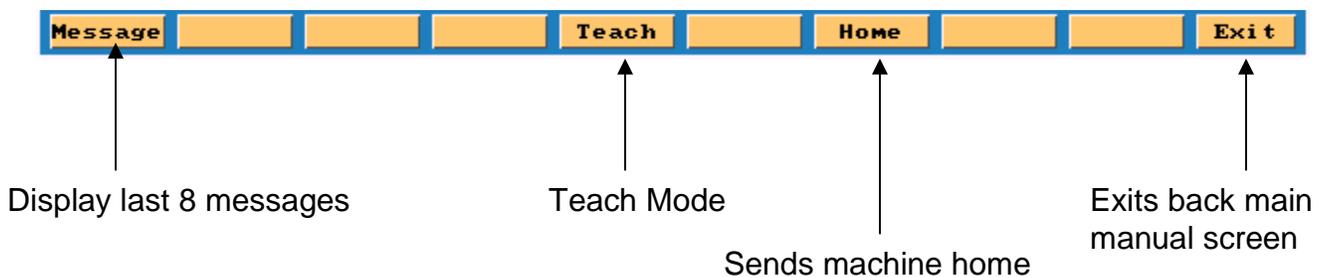
Alfa - Numeric key board

Note :- Most used key are _yellow_ . Most key also double functions , the shift key is use to use secondary functions.

Clear key

Space key

Cursor keys

Shift key

Enter key
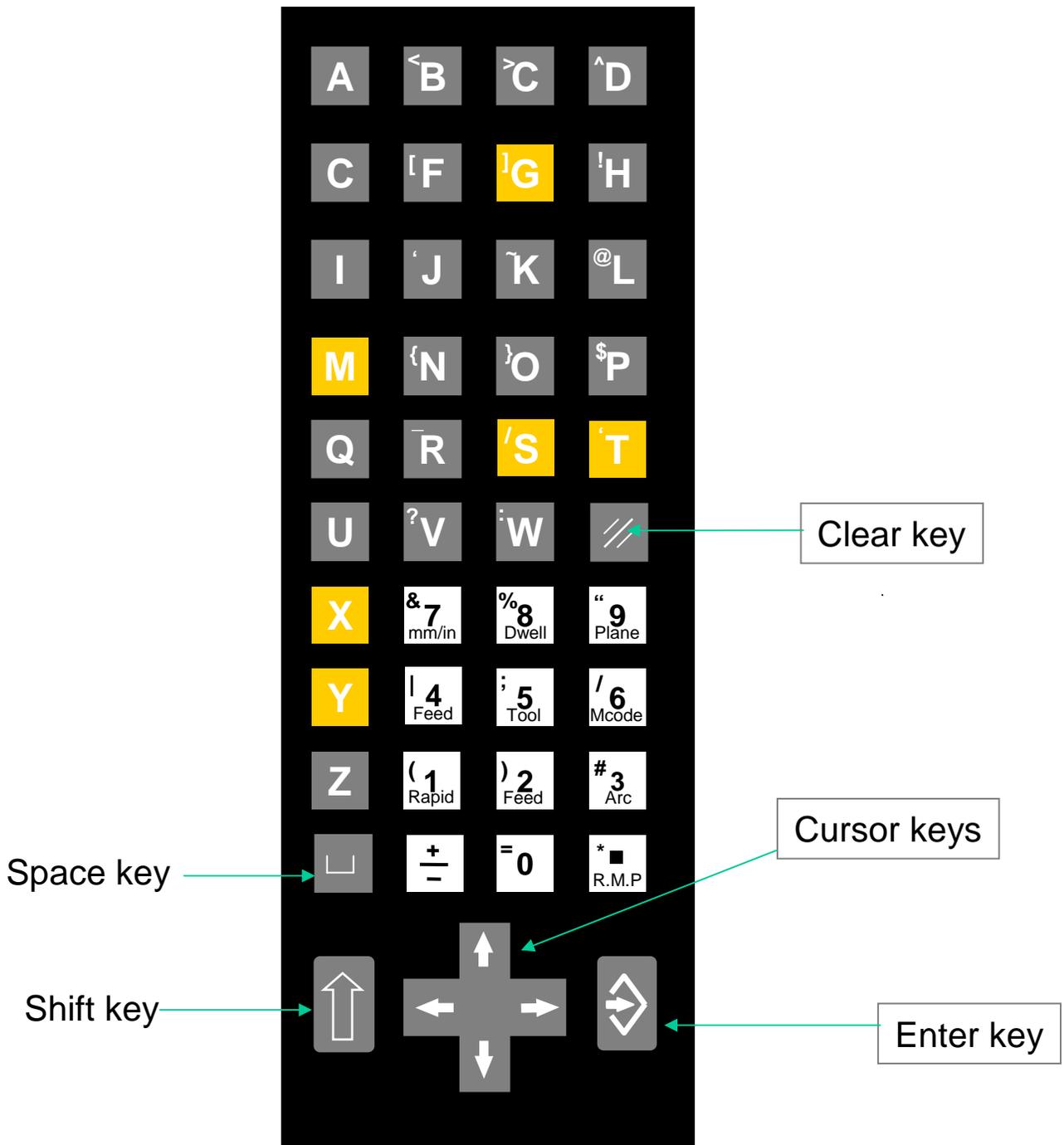
Manual panel

There are two type of Manual panels.
1. As shown below.
2. The other without Handwheel on left side

Manual spindle keys

Manual handwheel   Spindle override

Feedrate override

Servo reset

Axis feed type

Hold

Start

Axis select for handwheel   Direction key for manual

Emergence stop
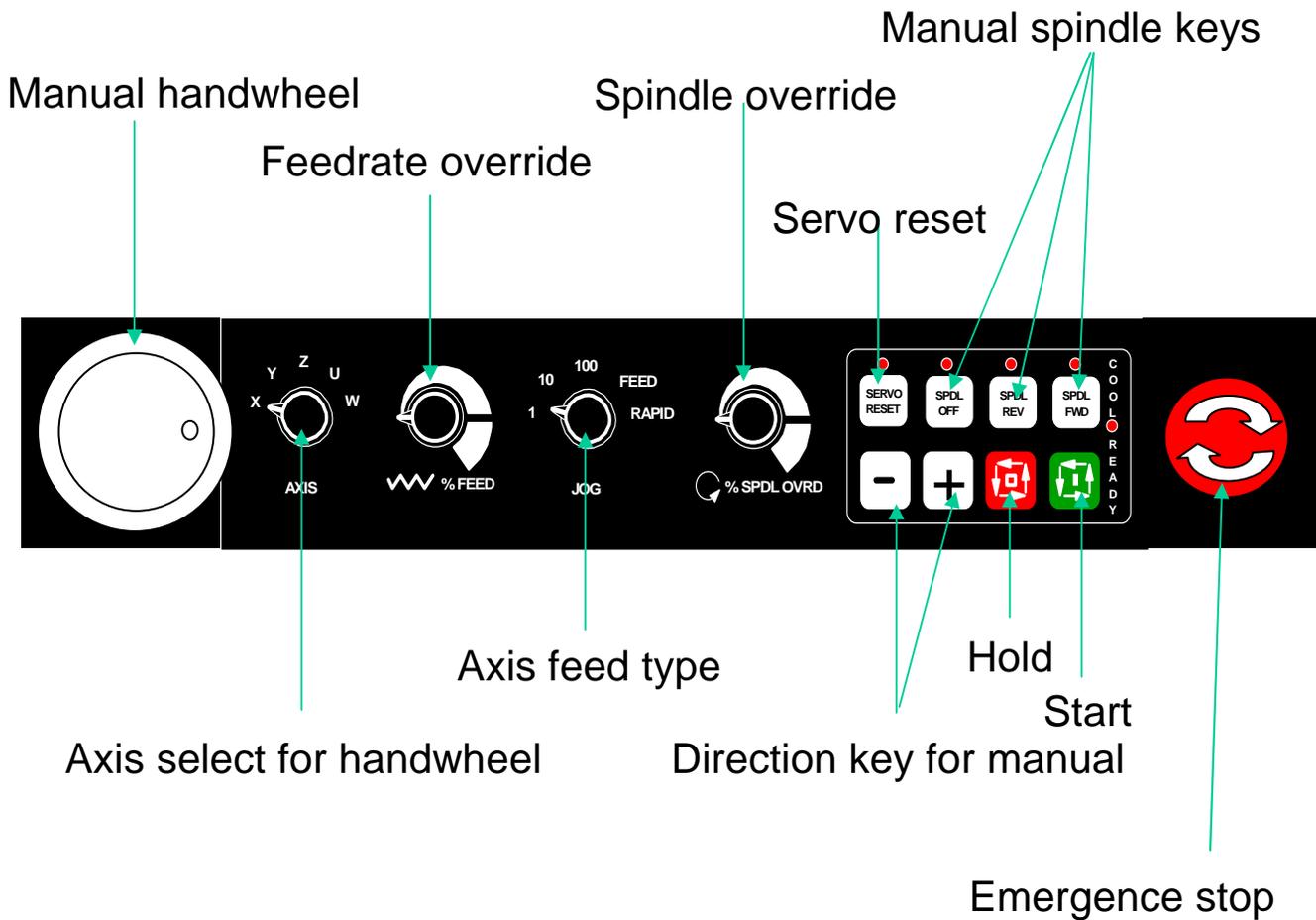
<u>MOVING WITH JOG MOVEMENTS</u>

1) Turn the jog mode selector switch on the **MANUAL PANEL** to **100, 10, 1, FEED or RAPID**

2) Select the axis to move with the **AXIS SELECTOR** switch at the **MANUAL PANEL**

3) Press the [−] or the [**+**] key to move in the desired direction.

<u>MOVING WITH COMMANDS</u>

Type commands as needed and press the START button [image] located on the MANUAL PANEL

EXAMPLE:

Type: **G0 G90 Z-5.** and press Start [image] Z axis goes to Z-5. in ABS and RAPID

EXAMPLE:

Type **G97 S1000 M3** and press START [image] to start the spindle at a fixed 1000 rpm.

MOST COMMON G-CODE COMMANDS TO REMEMBER:

G00:   Rapid move.

G01:   Feed move.

G02:   Arc clockwise

G03:   Arc counter clockwise

G17:   XY plane

G18:   XZ plane

G19:   Yz plane

G40:   Cutter comp off

G41:   Cutter comp left

G42:   Cutter comp right

G53 Ox: Fixture offsets (absolute shift from Machine Home).

G70:   Inch programming

G71:   MM programming

G80:   Used for **Drilling** Cycles.

G90:   Absolute co-ordinate system

G91:   Incremental co-ordinate system

G92:   Incremental shift

G94:   Feed Rate in Inch per minute (mm per minute).

G95:   Feed Rate in Inch per revolution (mm per revolution)

M0:   Stop program .

M2:   End of program

M3:   Spindle forward

M4:   Spindle reverse

M5:   Spindle OFF

M8:   Coolant ON

M9:   Coolant OFF

ADDRESS LETTERS USED IN PROGRAMMING

A: Used for angles around X axis.
B: Used for angles around Y axis.
C: Used for angles around Z axis.
D: Designates tool diameter in a program.
F: Feedrate.
I: Circle center X axis.
J: Circle center Y axis.
K: Circle center Z axis.
L: Dessignates tool length in a program.
M: Miscellaneous functions.
N: Line number prefix.
O: Program or subroutine number prefix.
P: Subroutine number call prefix.
S: R.P.M. prefix.
T: Tool call.
U: Rotary axis or spindle.
V: Rotary axis or spindle.
W: Rotary axis or spindle.
LOOP: Repete operation.
END: End of loop.

Note: Some of these letters are used inside canned cycles
           also some that are not listed above.
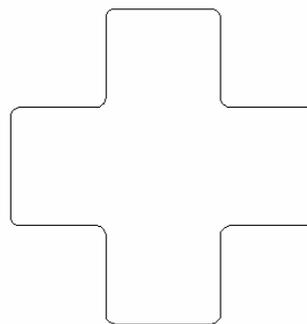
RETREIVING PROGRAMS FROM DISK

Transferring a program from a disk into the control

Programs can be restored into the control if they have been previously saved on a disk.
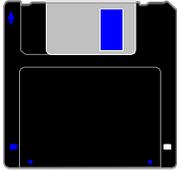
Insert the disk into the floppy disk drive (**A:**) on the machine.

From the MANUAL mode press **Program** then press **SHIFT**

Next, press **Log** select "**A:**" and press **ENTER**

Using the arrow keys hi-lite the program to be restored and press
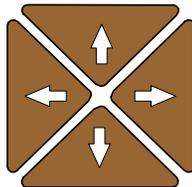
Utility then press twice to select **COPY** and then **C:**

WARNING

Remove the diskette when complete and save in a safe place. Do not leave the diskette in the machine, the will not start correctly with disk in the drive.

Keys to use when using Computer key board

FUNCTION	CNC KEYBOARD	COMPUTER KEYBOARD
X Axis Command	X	X Key
Y Axis Command	Y	Y Key
Z Axis Command	Z	Z Key
Preparity G Code	G	G Key
Machine Function Code	M	M Key
Spindle Function Code	S	S Key
Tool Command	T	T Key
ENTER		ENTER Key
SHIFT		SHIFT Key
Cycle HOLD Key		Alt + H Key
Cycle START Key		Alt + S Key
Cursor UP, DOWN, LEFT and RIGHT		ARROW Keys
CLEAR Key		Alt + C Key
FUNCTION Keys F1 - F10	F1	F1 - F10 Keys

Definitions of Function keys are given on screen

5000M CNC CONTROL
HELP MENU'S

The **HELP MENU'S** are access by pressing. **F1** HELP

This can be done from either Manual or Edit.

Manual mode soft keys

| Help | Program | Edit | Manual | S.Step | Auto | Delete | Insert | Tool | HandWl |

Edit mode soft keys

| Help | Del | Ins | DelBlk | PgUp | PgDn | Move | Editing | Misc | Exit |

First **Help** screen
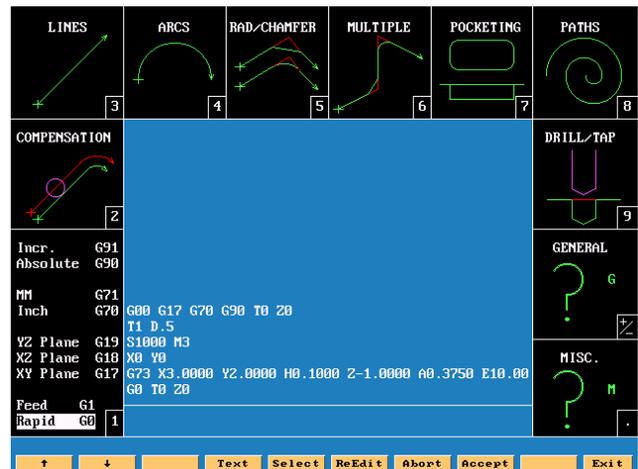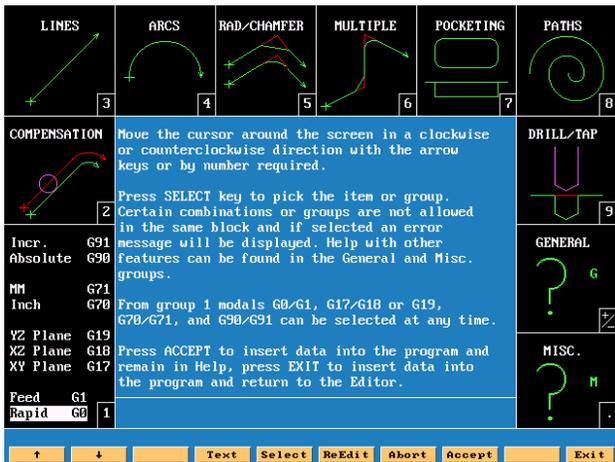
Note: The center of the screen. There are two different displays, one has text about **Help** the other shows the program as it is being entered.

Press **F4** Text will toggle these screens.

| ↑ | ↓ | | | Text | Select | ReEdit | Abort | Accept | | Exit |

| ↑ | ↓ | | | Select | | Abort | Accept | Prev | Exit |
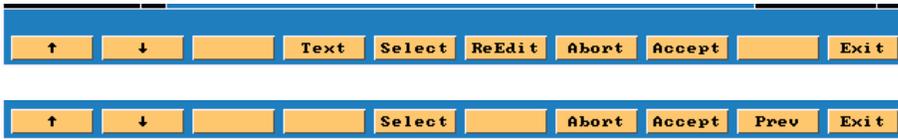
There are two different set of soft keys , as shown above.

| F1 | ↑ | Arrow up and around to desired selection. |

| F2 | ↓ | Arrow down and around to desired selection. |

| F3 | | Not used. |

| F4 | Text | Text changes center of screen to diplay program. |

| F5 | Select | Selects required cycle.Enter will as do this. |

| F6 | ReEdit | ReEdits a cycle after it is already in a program. |

| F7 | Abort | Goes back to edit without saving last set of inputs. |

| F8 | Accept | Accepts inputs and stays in HELP. |

| F9 | Prev | Goes back to previous page in HELP menu. |

| F10 | Exit | Exits to EDIT page and saves inputs. |

Section as shown below these are the major defaults. Hight lite the required in put and

press either **Select** or enter.


```
Incr.      G91  i
Absolute   G90  m
                f
MM         G71  g
Inch       G70  F
                G
YZ Plane   G19
XZ Plane   G18  P
XY Plane   G17  r
                t
Feed    G1
Rapid   G0   1
```

Use up arrow key to move hight to next selection.

Press the number 2 key or arrow up to number 2 .

Press either **Select** or enter
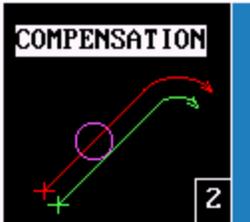
Note:- the reverse color around Compensation.

Press either **Select** or enter
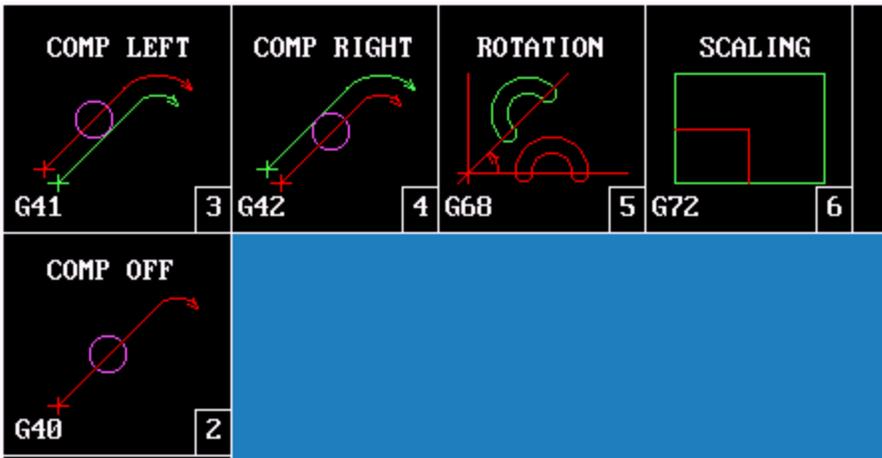
1. Puts a **G40** into program comp off.
2. Enters **G41** into program cutter comp left.
3. Enters **G42** into program cutter comp right.
4. G68 rotates a shape around a center.
5. G72 scale program to required size.

Rotation G68

```
              G68 - ROTATION

      NOTE: G68 alone, cancels rotation.
            If P and L used, S must be used.

                        Rotation ctr.I  [      ]  ◄─── Center of rotation X axis.
                        Rotation ctr.J            ◄─── Center of rotation Y axis.
                        First Angle  S            ◄─── Start angle when using loop.
                        Angle        C +0.00000   ◄─── Angle between loops or single shot.
                        Subprogram # P            ◄─── Subroutine #.
                        Num of Times L            ◄─── Number of repeats.



                             0 Degrees.


              Rotation Center
```
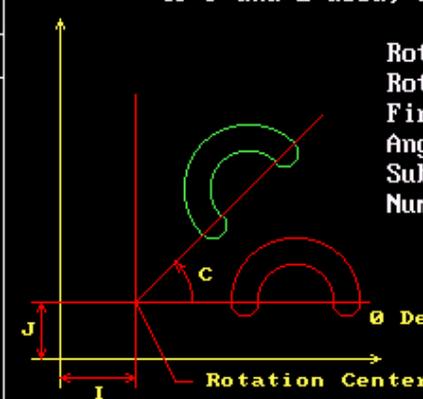
Note: The only entry that has to be program is **C** because it has 0 (Zero's) next to it.

This cycle can be programmed in main program or in a subroutine. If programmed in the main, the cycle is entered and then the diminsion of the shape and turned off with a **G68**.

```
G00 G17 G70 G90 T0 Z0
T1
S1000 M3
X5 Y4
G68 I5.0000 J4.0000 C45.00000
G0 X7 Y4 Z.1
G1 Z-.2
X9
Z.1
G68
G0 T0 Z0
X-1 Y-1
M2
```

This example is show rotation just one time not using a subroutine, note the G68 to turn off rotation.

```
G00 G17 G70 G90 T0 Z0
T1
S1000 M3
X5 Y4
G68I5.0000J4.0000S0.00000C45.00000P1L8
G0 T0 Z0
X-1 Y-1
M2

O1
G0 X7 Y4 Z.1
G1 Z-.2
X9
Z.1
M99
```

Example on left show rotation using subroutine, note there is no **G68** turning off rotation ,it's not required when programming this way.

Scaling G72

```
                    G72 SCALING

         NOTE: G72 alone, cancels scaling.

                              Scale factor X
   2                          Scale factor Y
                              Scale factor Z
  91
  90

  71
  70

  19
  18
  17
```
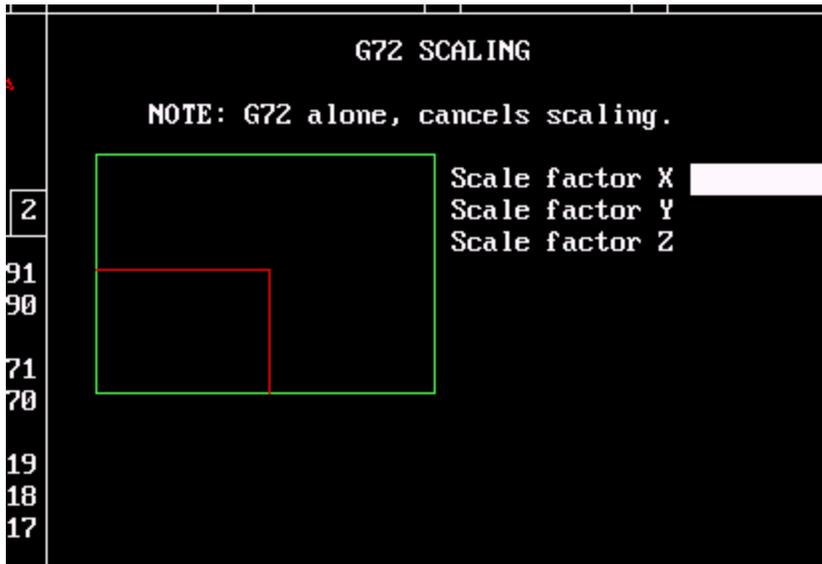
When using scaling if threre are any the axis must be scaled the same
on both of these axis.If part is required to be half size .5 would be factor.
G72 alone will turn off scaling.

Lines

Press # 3 either **Select** or enter.
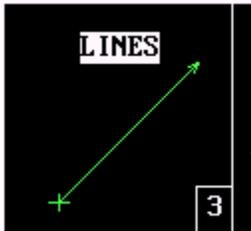
Screen will now appear as below.

Inputs will change according to which plane is active.
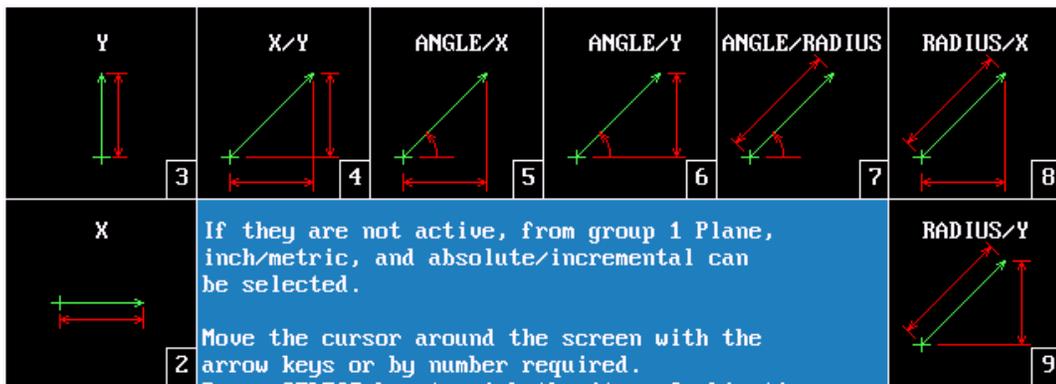
2. X axis input only.

3. Y axis input only.

4. X and **Y** axis.

5. Angle and **X** axis.

6. Angle and **Y** axis.

7. Angle and **Radius**

8. Radius and **X** axis

9. Radius and **Y axis**

Arc's
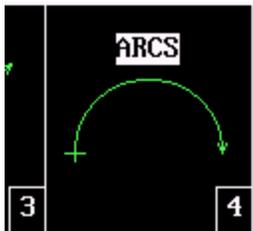
Press # 4 either **Select** or ➡ enter.

Screen will now appear as below.
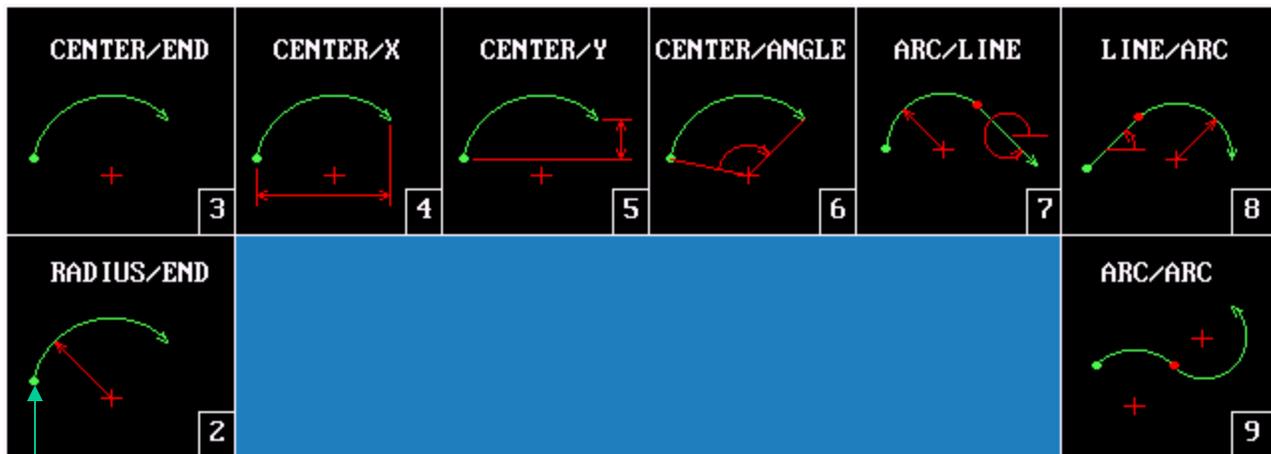
Tool must be at start point before inputting arc's
Centers of arcs X=I , Y=J and Z=K
Inputs will change according active plane.

2. Radius and **End Point.**

3. Center and **End Point.** This can be used for helical interpolation (thead mill.)

4. Center and **X End Point.**

5. Center and **Y End Point.**

6. Center and **Angle.** Angle is dependant Absolute or increamental.

7. Arc and **Line.** Inputs are Radius, Angle and End Point X and Y

8. Line and **Arc.** Inputs are Angle, Radius and End Point X and Y

9. Arc and **Arc.** Inputs Center X and Y first arc, Center X and Y second arc
and End Point X /Y

Corner Rounding And Chamfering

Press # 5 either **Select** or enter.

Screen will now appear as below.
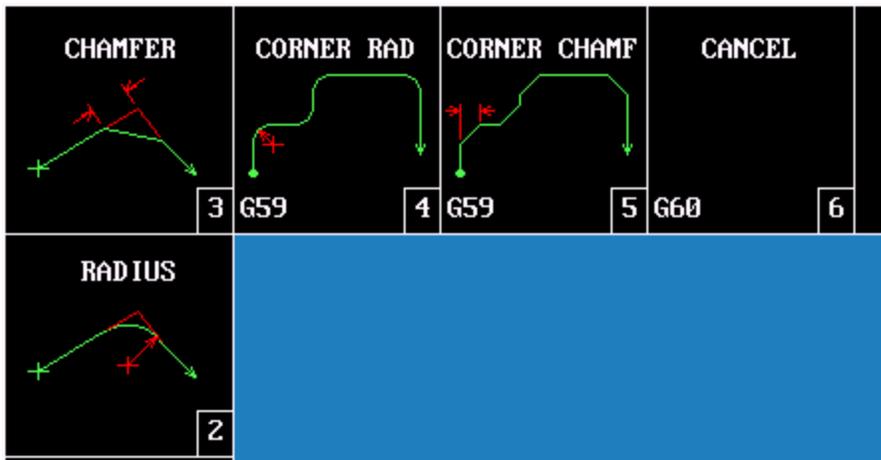
2. Radius   One shot corner rounding.Inputs mid point X / Y,radius
                        and end point X/Y.
3.Chamfer   One shot chamfer. Inputs mid point X/Y, chamfer and
                        end point X/Y.
4.Corner Radius   Modal command puts radius on all intersects.

5.Corner Chamfer Modal command puts chamfer on all intersects.

6.Cancel   Cancels #4 and #5

Multiple line, arc and chamfer moves.

MULTIPLE

6
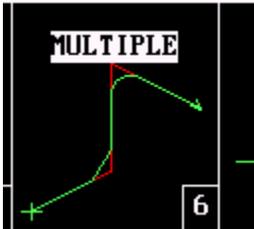
Press # 6 either **Select** or → enter.

Screen will now appear as below.

RADIUS	CHAMFER	RAD/RAD	CHAMF/CHAMF	RAD/CHAMF	CHAMF/RAD
3	4	5	6	7	8

DEFINITION

2
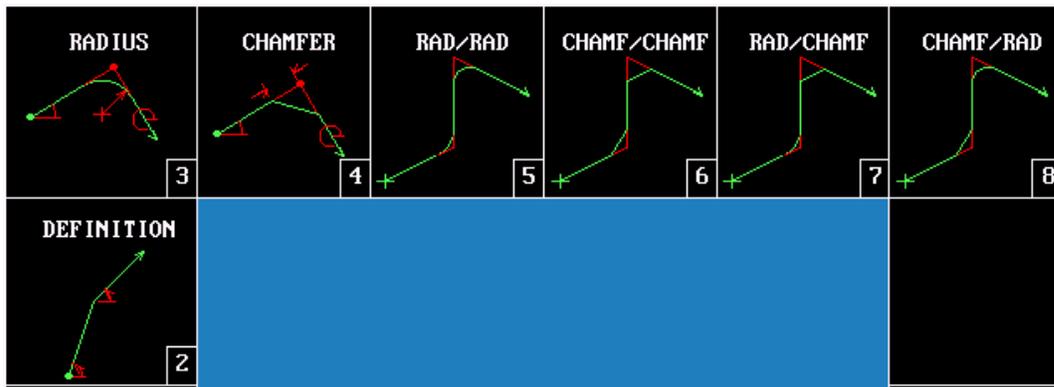
Tool must be positioned at start point.
Inputs will change according to active plane.

2. Definition   Inputs first angle, second angle and end point.

3.Radius   Inputs first angle, radius, second angle and
end point.

4.Chamfer   Inputs first angle, radius, second angle and
end point.

5.Rad/Rad   Inputs first angle, first radius, second angle,
mid point, second radius and end point.

6.Chamf/Chamf Inputs first angle, first radius, second angle,

7.Rad/Chamf   Inputs first angle, radius, second angle, mid point
X/Y chamfer and end point.

8.Chamfer/Rad Inputs first angle, chamfer, second angle, mid point
X/Y, radius and end point.

Pockets

Press # 7 either **Select** or enter.
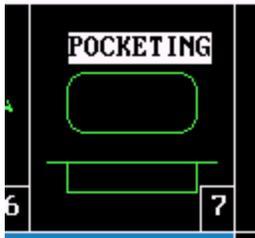
Screen will now appear as below.

Cutter comp is built into all pocket except Mold Rotation.
X and Y centers are Optional but if not entered will assume it is positioned at center of pocket.

#2. Frame Milling.	Leave island in middle of pocket.
#3. Hole Milling.	Enlarges existing hole, used on smaller holes.
#4. Circular Pocket.	Cuts flat bottom circular pocket.
#5. Rectangular Pocket.	Cuts flat bottom rectangular pocket.
#6. Area Clearance.	Cuts irregular shape pocket and takes profile cut.
#7. Mold Rotation.	Rotates a profile around an axis.
#8. Elbow Milling.	Produces a radial groove.
#9. Draft Pocket.	Rectangular pocket with angled sides.
+/–. Plunge pockets.	Rectangular and circular pocket plunging straight down.

Plunge Pockets

#2. Circular Plunge Pocket	Plunges straight into material.
#3. Rectangular Plunge Pocket.	Plunges straight into material.

Frame pocket G75

Only the input with in Zeros have to be entered the rest are optional. It will assume being at the center of pocket if no dimension are entered.

```
              G75 FRAME MILLING
   Neg. stepover = CW.   X Center    X
                         Y Center    Y
                         Length      M  +0.0000
                         Width       W  +0.0000
                         Start Hgt.  H  +0.0000
                         Z Depth(abs) Z +0.0000
                         Stepover    A
                         Max. Z cut  B
                         Ramp Feed.  I
                         Rough Feed. J
                         Inside Rad. U  +0.0000
                         Outside Rad. V
                         Frame Width C  +0.0000
                         Finish Stock S
                         Finish Feed. K
                         Retract Hgt. P
```
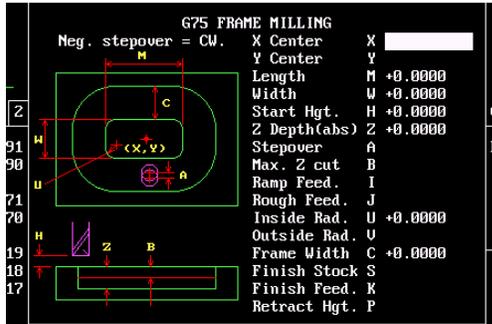
X Center	X	Center of pocket X axis.If not entered will assume tool is at center of pocket.Optional
Y Center	Y	Center of pocket Y axis.If not entered will assume tool is at center of pocket.Optional
Length	M	Length of island (X).
Width	W	Width of island (Y).
Start Hgt.	H	Start height .1inch or 2mm above top surface of pocket.
Z Depth (abs)	Z	Absolute depth to bottom of pocket.
Stepover	A	Cut per pass, not to exceed 70% of cutter dia. Negative value path will climb mill.
Max. Z cut	B	Depth per pass Z axis.Optional
Ramp Feed	I	Feedrate when feeding down into pocket. Optional
Rough Feed	J	Feedrate roughing pocket.Optional
Inside Rad.	U	Radius on corners of island.
Outside Rad.	V	Radius on outside, will assume cutter radius if no entry. Optional
Frame Width	C	Dimension from island to outside.
Finish Stock	S	Amount of material left for finish pass.Optional
Finish Feed	K	Feedrate for finish pass.Optional
Retract Hgt.	P	High retract allows tool to be move above the surface part when finished. Optional

Hole Milling G76

A good use for this cycle to produce small counterbores.

```
        G76 HOLE MILLING

     NOTE: Negative diameter = CW.

        X Center    X
        Y Center    Y
        Diameter    D  +0.0000
        Z Depth(abs) Z
        Z Max. cut  B
        Z Start Hgt. H
        Finish Stock S
        Rough Feed. J
        Finish Feed. K
```
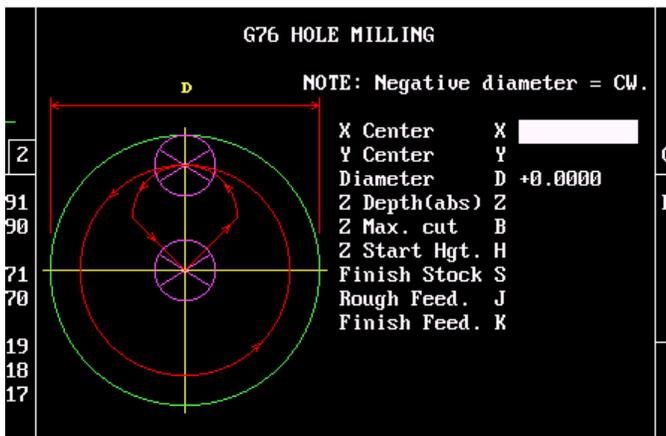
Diameter	D	Diameter of pocket.
Rough Feed	J	Feedrate roughing.
Finsih Stock	S	Finish Stock.
Finish Feed	K	Feedrate finish pass.

Circular pocket G77

```
        G77 CIRCULAR POCKET MILLING

              NOTE: Negative diameter = CW.

        X Center      X
        Y Center      Y
        Z Start Hgt.  H +0.0000
        Z Depth(abs)  Z +0.0000
        Diameter      D +0.0000
        Stepover      A
        Z Max. cut    B
        Rough Feed.   I
        Finish Stock  S
        Finish Feed.  K
        Retract Hgt.  P
```

X Center	**X**	Center X axis.	Optional
Y Center	**Y**	Center Y axis.	Optional
Z Start Hgt.	**H**	Start height .1 above surface to be cut into.	
Z Depth (abs)	**Z**	Absolute Z depth.	
Diameter	**D**	Diameter of pocket, comp built in.	
Stepover	**A**	Move over per pass.	Optional
Z Max. cut	**B**	Max. Z depth per pass.	Optional
Rough Feed.	**I**	Feedrate roughing.	Optional
Finish Stock	**S**	Amount of material left for finish pass.	Optional
Finish Feed.	**K**	Finish pass feedrate.	Optional
Retract Hgt.	**P**	High returnwhen finished.	Optional

Rectangular Pocket. G78

```
        G78 RECTANGULAR POCKET MILLING
        Neg. stepover = CW.

        X Center      X
        Y Center      Y
        Length        M +0.0000
        Width         W +0.0000
        Z Start Hgt.  H +0.0000
        Z Depth(abs)  Z +0.0000
        Corner Rad.   U
        Stepover      A
        Z Max. Cut    B
        Ramp Feed.    I
        Rough Feed.   J
        Finish Stock  S
        Finish Feed.  K
        Retract Hgt.  P
```

X Center	**X**	Center of pocket X
Y Center	**Y**	Center of pocket Y
Length	**M**	Actual length X axis
Width	**W**	
Z Start Hgt.	**H**	Actual width Y axis
Z Depth (abs)	**Z**	.1 above surface to be cut
Corner Rad.	**U**	Absolute depth of pocket
Stepover	**A**	Radius in corners
Z Max. Cut	**B**	70% of cutter or less
Ramp Feed	**I**	Max depth per pass
Rough Feed	**J**	Feedrate on 3 axis first move
Finish Stock	**S**	Feedrate for roughing
Finish Feed	**K**	Amount of stock for finish cut
Retract Hgt.	**P**	Finish feedrate
		Retract after finished.

Area Clearance   G169.

```
        G169 AREA CLEARANCE

              Input Sub. # W  +0
              X            X
              Y            Y
              Z Start Hgt. H  +0.0000
              Z Depth(abs) Z  +0.0000
              Cut Angle    C
              X Start      D
              Y Start      E
              Stepover     A  +0.0000
              Z Max. Cut   B
              Ramp Feed.   I
              Rough Feed.  J
              Finish Stock S
              Finish Feed. K
              Retract Hgt. P
```

Input Sub #	W	Subroutine number.
X	X	X position tool will Z down into part
Y	Y	Y position tool will Z down into part
Z Start Hgt.	H	Start height .1 above top of pocket
Z Depth (abs)	Z	Total depth of pocket absolute
Cut Angle	C	Used if starting in middle of radius
X Start	D	Position of cut at start X axis. Optional
Y Start	E	Position of cut at start Y axis. Optional
Stepover	A	Cutter stepover each pass
Z Max. Cut	B	Max depth of cut per pass.
Ramp Feed	I	Ramp feedrate Z down
Rough Feed	J	Rough feedrate
Finish Stock	S	Stock left for finish pass
Finish Feed	K	Finish feedrate
Retract Htg.	P	Retract after finished.

Mold Rotation G45

```
         G45 MOLD ROTATION

  NOTE: Two subprograms required.
    J is Z CL if Axis of Rotation is X or Y.
    J is Y CL if Axis of Rotation is Z.
    K only used if Axis of Rotation is X or Y.

              Start Angle    A  +0.00000
              End Angle      B  +0.00000
              Num of Cycles  C  +0
              Num of Fwd Sub F  +0
              Num of Rev Sub R  +0
              Axis of Rotation X
              Other Ax. CL   I
              CenterLine     J
              Rotation Angle K
```
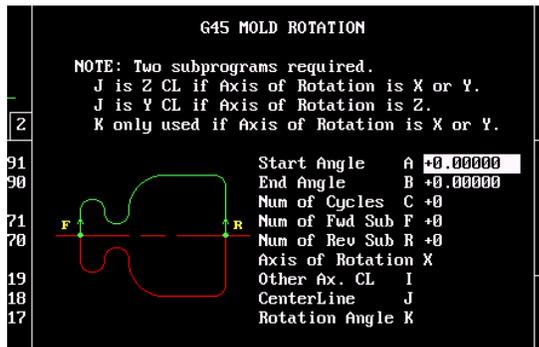
Start Angle	A	Angle where rotation is going to start
End Angle	B	Angle where rotation is going to end
Num of Cycles	C	1 cycle equals 1 Fwd and 1 Rev. Sub.
Num of Fwd Sub	F	Sub. Profile forward direction
Num of Rev Sub	R	Sub. Profile Reverse direction
Axis of Rotation	X	Axis rotation is around X,Y or Z
Other axis CL	I	Center line X or Y if not Zero
Centerline	J	Center line Z Axis if not Zero
Rotation Angle	K	Angle rotation Z axis only

14

Elbow Milling G49

```
          G49 ELBOW MILLING
NOTE: Tool must be at starting position.
          Start Radius   B +8.0000
          End Radius     K +0.0000
          Incl. Angle    A +0.00000
          Num of Cycles  C +0
          X Center       I
          Y Center       J
          Direction CCW+ D
          End Angle      F
          Start Angle    E
          Ctr.line Rad.  R
          Rapid Height   Z
          Start Height   H
          Rough Feed.    U
          Finish Stock   S
          Finish Feed.   V
```

Start Radius	B	Radius at start of Elbow
End Radius	K	Radius at end of Elbow
Included Angle	A	Included angle
Num of Cycles	C	Sub. Profile forward direction
X Center	I	Sub. Profile Reverse direction
Y Center	J	Axis rotation is around X,Y or Z
Direction CCW+	D	Center line X or Y if not Zero
End Angle	F	Center line Z Axis if not Zero
Start Angle	E	Angle rotation Z axis only
Ctr. Line Radius	R	Radius at center of elbow
Rapid Height	Z	Starting hieght above surface.
Start Height	H	Z height to start
Rough Feed	U	Rough feedrate
Finish Stock	S	Amount of stock for finish pass
Finish Feed	V	Feedrate for finish cut

Draft Pocket G73

```
        G73 DRAFT POCKET MILLING
NOTE: Position tool at ctr of lwr left corner rad.
        Length, bottom X +0.0000
        Width, bottom  Y +0.0000
        Start Hgt.     H +0.0000
        Z Depth(abs)   Z +0.0000
        Lower Left Rad A +0.0000
        Lower Rght Rad B
        Upper Rght Rad C
        Upper Left Rad D
        Draft Angle    E +0.00000
        Z step Rough   I +0.0000
        Max XY Stepovr V
        Finish Stk XY  S
        Z step Finish  Q
        Finish Feed.   R
        Flat 0, Ball 1 W
```

Note: Tool must be positioned at center of radius bottom left corner. Center of pocket must be cleared before using this cycle. When using flat endmill will go to programmed depth. If ball endmill uses will only go to depth minus cutter radius, .5 mill Absolute depth -1 actual depth it would go to is -.75.

Length, bottom	X	Length at bottom of pocket.	required
Width, bottom	Y	Width at bottom of pocket.	required
Start Height	H	Height above part to rapid.	required
Z Depth (abs)	Z	Absolute depth	required
Lower Left Rad.	A	Lower left Radius	required
Lower Right Rad.	B	Lower right radius	optional
Upper Left Rad.	C	Upper left radius	optional
Upper Right Rad.	D	Upper right radius	optional
Draft Angle	E	Draft angle Degrees	required
Z step Rough	I	Depth per pass in Z axis	required
Max XY Stepover	V	Maximum stepover XY	optional
Finish Stock XY	S	Finish stock XY	optional
Z step Finish	Q	Z step finish pass	optional
Finish Feed	R	Finish feedrate	optional
Flat 0, Ball 1	W	Flat mill = 0 Ball mill = 1	optional

PLUNGE POCK.

High lite plunge pockets press

RECT. POCK.

G178   3

CIRC. POCK.

G177   2

Plunge Circular Pocket G177

G177 PLUNGE CIRCULAR POCKET MILLING
NOTE: Position tool at center (or 3 o'clock)
of pocket, Neg. diameter = CW.

X Center   X
Y Center   Y
Z Start Hgt. H +0.0000
Z Depth(abs) Z +0.0000
Diameter   D +0.0000
Stepover   A
Z Max. cut   B
Z Feedrate   I
Rough Feed. J
Finish Stock S
Finish Feed. K
Retract Hgt. P
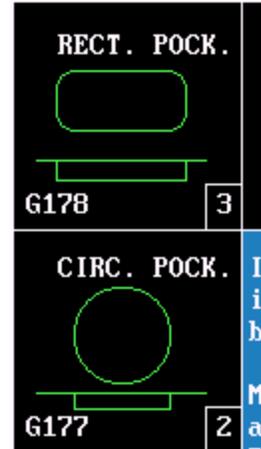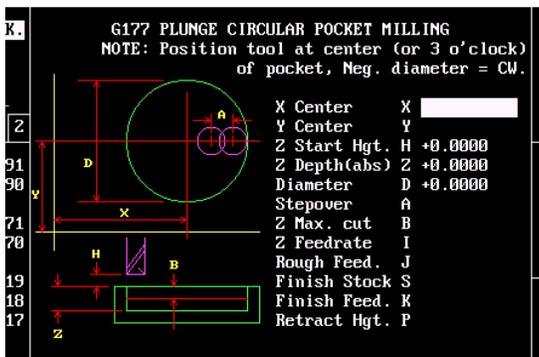
X Center	**X**	Center X axis	Optional
Y Center	**Y**	Center Y axis. Same as above.	Optional
Z Start Hgt.	**H**	Start height .1 above surface to be cut.	
Z Depth (abs)	**Z**	Absolute Z depth.	
Diameter	**D**	Diameter of pocket, comp built in.	
Stepover	**A**	Move over per pass.	Optional
Z Max. cut	**B**	Max. Z depth per pass.	Optional
Z Feedrate	**I**	Feedrate plunging	Optional
Rough Feed.	**J**	Feedrate roughing.	Optional
Finish Stock	**S**	Amount of material left for finish pass.	Optional
Finish Feed.	**K**	Finish pass feedrate.	Optional
Retract Hgt.	**P**	High returnwhen finished.	Optional

Plunge Pocket Pocket G178
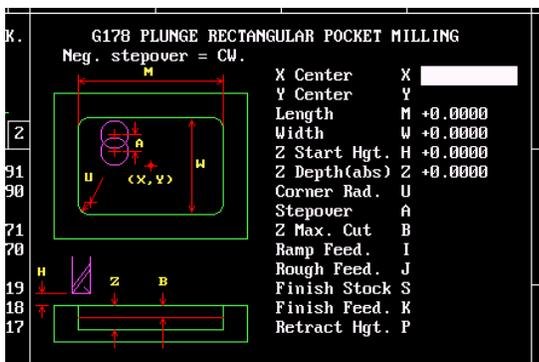
G178 PLUNGE RECTANGULAR POCKET MILLING
Neg. stepover = CW.

X Center   X
Y Center   Y
Length   M +0.0000
Width   W +0.0000
Z Start Hgt. H +0.0000
Z Depth(abs) Z +0.0000
Corner Rad. U
Stepover   A
Z Max. Cut   B
Ramp Feed.   I
Rough Feed. J
Finish Stock S
Finish Feed. K
Retract Hgt. P

X Center	**X**	Center of pocket X
Y Center	**Y**	Center of pocket Y
Length	**M**	Actual length X axis
Width	**W**	Actual width Y axis
Z Start Hgt.	**H**	.1 above surface to be cut
Z Depth (abs)	**Z**	Absolute depth of pocket
Corner Rad.	**U**	Radius in corners
Stepover	**A**	70% of cutter or less
Z Max. Cut	**B**	Max depth per pass
Ramp Feed	**I**	Feedrate on 3 axis first move
Rough Feed	**J**	Feedrate for roughing
Finish Stock	**S**	Amount of stock for finish cut
Finish Feed	**K**	Finish feedrate
Retract Hgt.	**P**	Retract after finished.

Pockets with Islands (G162)

Format: G162 An Bn Cn Dn En

This cycle allows islands in irregular pockets. The main pocket must the lowest subroutine number. Normally, this would be one (1). Pockets with Islands can be programmed using:

- DXF (see "Section 17, Using DXF for Pockets with Islands (G162)")
- CAM (see "Section 18, Example #12 Using CAM for Pockets with Islands (G162)")
- Subroutines

More than one **G162** Island cycle can be programmed at a time. They may be strung together, but on separate lines. Islands can be programmed inside of islands. Five islands can be put on a line. The shape number subroutine number is used as inputs.. Refer to
Table 5-21.

Activate a tool prior to programming G78, so cutter diameter is known.

Table 5-21, G162 Address Words

Address Word	Description
A	First island. Required.
B	Second island. Optional.
C	Third island. Optional.
D	Fourth island. Optional.
E	Fifth island. Optional.

Using Subroutines for Pockets with Islands

The program below is the same one used in the DXF portion with subroutines added for the letters. In the third G162 some of the numbers have negative sign (-) in front of them, this changes the side of the cutter comp for the islands in islands. See **Figure 5-13, Subroutines Pockets with Islands Example Workpiece** and **Table 5-22, Pockets with Islands Subroutines Programming Example**.
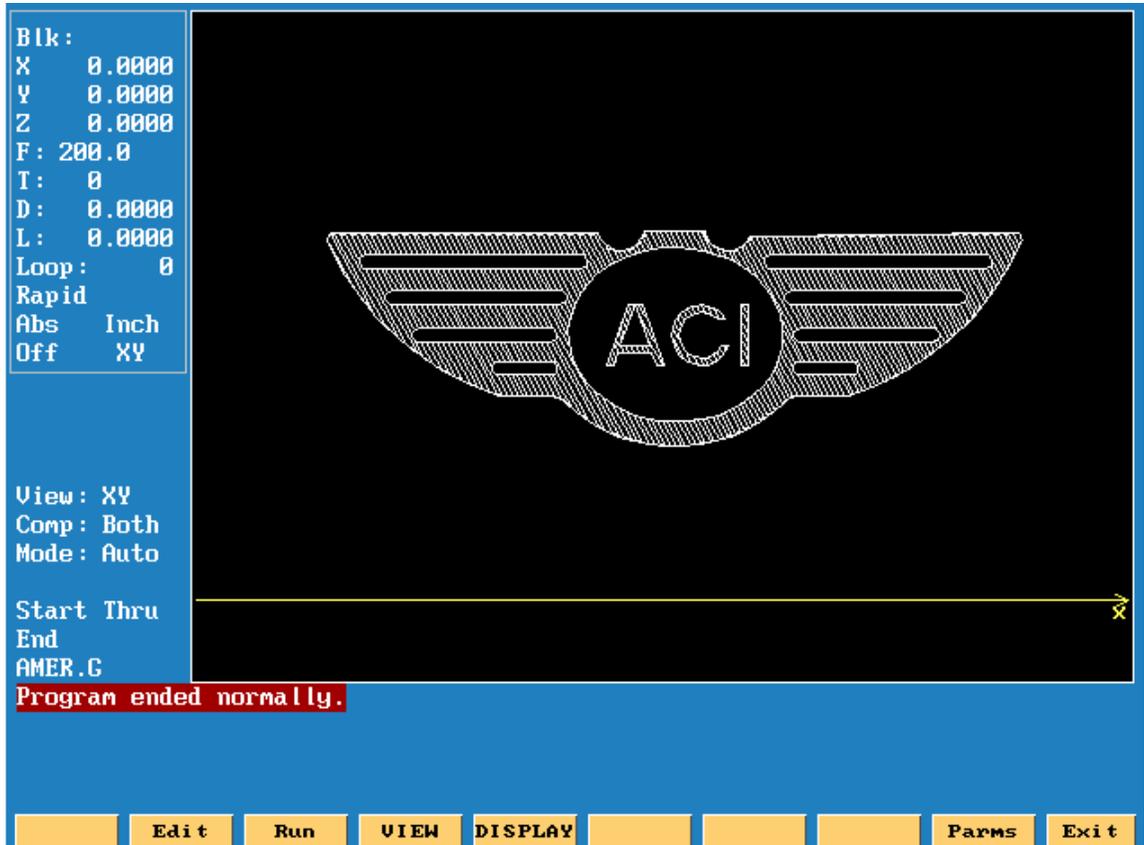
Figure 5-13, Subroutines Pockets with Islands Example Workpiece

Table 5-22, Pockets with Islands Subroutines Programming Example

N1	G00 G17 G70 G90
N2	T1D.0205 L-1 M6
N3	S1250M3
N4	M8
N5	G53O01
N6	G162 A2 B3 C4 D5 E6
N7	G162 A7 B8 C9 D10
N8	G162 A-11 B12 C-13 D-14
N9	G169 W1 H0.1 Z-0.0050 C299. A0.0080 I5.0 J12.0 S0.0010 K10.0 P1
N10	M2
N11	
N12	O11
N13	X1.1044 Y0.5
N14	Z0.1
N15	G01 Z-0.005
N16	G01 X1.159 Y0.65
N17	X1.2052

N18	X1.2598 Y0.5
N19	X1.2226
N20	X1.2135 Y0.525
N21	X1.1507
N22	X1.1416 Y0.5
N23	X1.1044
N24	G00 Z0.1
N25	M99
N26	
N27	
N28	O12
N29	X1.1634 Y0.56
N30	Z0.1
N31	G01 Z-0.005
N32	G01 X1.1821 Y0.6112
N33	X1.2007 Y0.56
N34	X1.1634
N35	G00 Z0.1
N36	M99
N37	
N38	O13
N39	X1.4007 Y0.55
N40	Z0.1
N41	G01 Z-0.005
N42	G01 X1.3612
N43	G02 Y0.6 I-0.0312 J0.025
N44	G01 X1.4007
N45	G03 Y0.55 I-0.0707 J-0.025
N46	G00 Z0.1
N47	M99
N48	
N49	O14
N50	G0 X1.42Y.5
N51	G1 X1.42Y.65
N52	G1 X1.460 Y.65
N53	G1 X1.460 Y.5
N54	G1 X1.42 Y.5
N55	M99

#2. Ellipse.	Produces an ellipse, uses special cutter comp.
#3. Spiral.	Will cut a tapered thead.
#5. Facing.	Faces large surfaces.
#7. Circular Profile	Cut circle either inside or outside.
#8. Rectangular Profile.	Cut rectangle inside or outside.

Ellipse G05

Note: All dimensions are INCREMENTAL.
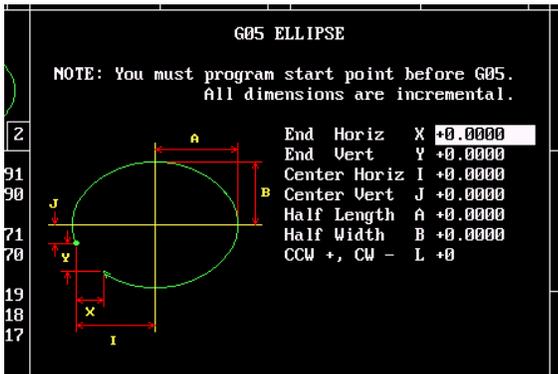
```
                    G05 ELLIPSE

    NOTE: You must program start point before G05.
          All dimensions are incremental.

              A         End  Horiz   X +0.0000
                        End  Vert    Y +0.0000
                        Center Horiz I +0.0000
                     B  Center Vert  J +0.0000
      J                 Half Length  A +0.0000
                        Half Width   B +0.0000
                        CCW +, CW -  L +0
        Y
          X
        I
```

End Horizontal	**X**	Distance from start to end X axis.
End vertical.	**Y**	Distance from start to end Y axis.
Center Horizontal	**I**	Distance to center from start.
Center Vertical	**J**	Distance to center from start.
Half lenght	**A**	Half length of ellipse x axis
Half width	**B**	Half width of ellipse Y axis.
CCW +,CW -.	**L**	Direction of cut.

Cutter comp for ellipse uses **M1040** X0 = off, X1 = outside and X2 = inside.
Cutter must be positioned in compensated position before ellipse is programmed.

Spiral G06

Note: All dimensions are INCREMENTAL.

```
              G06 SPIRAL (AND HELIX)

    NOTE: You must program start point before G06.
          All dimensions are incremental.
        X
                    L   End  Horiz   X +0.0000
                        End  Vert    Y +0.0000
                        End Depth    Z +0.0000
                     J  Center Horiz I +0.0000
      Y                 Center Vert  J +0.0000
                        Number revs. L +0.0
                          CCW = +, CW = -

          I
```

End Horizontal	**X**	Distance from start to end X axis.
End vertical.	**Y**	Distance from start to end Y axis.
End Depth Z	**Z**	Distance from start to end Z axis.
Center Horizontal	**I**	Distance to center from start.
Center Vertical	**J**	Distance to center from start.
Number of revolution	**L**	Number of turn it will make.
CCW +,CW -.		Direction of cut.

No compensation available for spiral.

Facing G170

```
                G170 FACING

    NOTE: Must specify A or B but not both.


                  Length       X +0.0000
                  Width        Y +0.0000
                  X Stepover   A
                  Y Stepover   B
                  Feedrate     F
    (X,Y)         Z Start Hgt. H +0.0000
                  Z Depth(abs) Z +0.0000
                  X Start      D
                  Y Start      E

          X
```
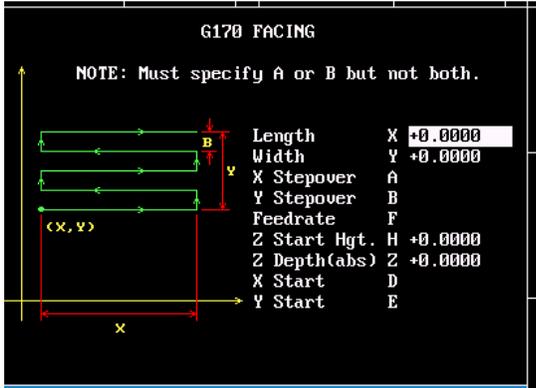
Length	**X**	Increamental length X axis.
Width	**Y**	Increamental width Y axis.
X Stepover	**A**	Stepover X .
Y Stepover	**B**	Stepover Y.
Feedrate	**F**	Feedrate.
Z Start hieght	**H**	Start height .1 above surface.
Z Depth Absolute	**Z**	Finish depth.
X Start	**D**	Start X axis.
Y Start	**E**	Start Y axis.

Note: Only A or B not both can be used.
Cutter will step away from start corner
by half the cutter diameter.

Circular profile G171
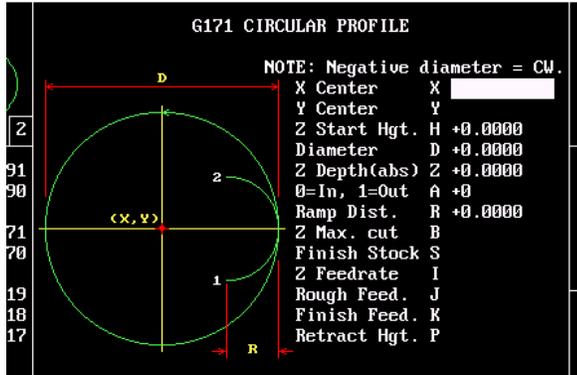
```
     G171 CIRCULAR PROFILE

     NOTE: Negative diameter = CW.
     X Center     X
     Y Center     Y
     Z Start Hgt. H +0.0000
     Diameter     D +0.0000
     Z Depth(abs) Z +0.0000
     0=In, 1=Out  A +0
     Ramp Dist.   R +0.0000
     Z Max. cut   B
     Finish Stock S
     Z Feedrate   I
     Rough Feed.  J
     Finish Feed. K
     Retract Hgt. P
```

Center X.	X	X Center (optional).
Center Y.	Y	Y Center (optional).
Start height.	H	Start height above surface to be cut.
Diameter.	D	Diameter of pocket (actual).
Z depth (absolute).	Z	Depth to be cut (absolute).
0 = inside, 1 = outside.	A	0 insde ,1 outside of circle.
Ramp Distance.	R	Size of ramp on radius.
Z Maximum cut.	B	Maximun depth in Z per pass.
Finish Stock.	S	Amount of stock left for finish cut.
Z Feedrate.	I	Feedrate in Z axis.
Rough feedrate.	J	Feedrate for roughing.
Finish Feedrate.	K	Feedrate for finishing.
Retract height.	P	High retact if higher than H value.

Rectangular Profile G172
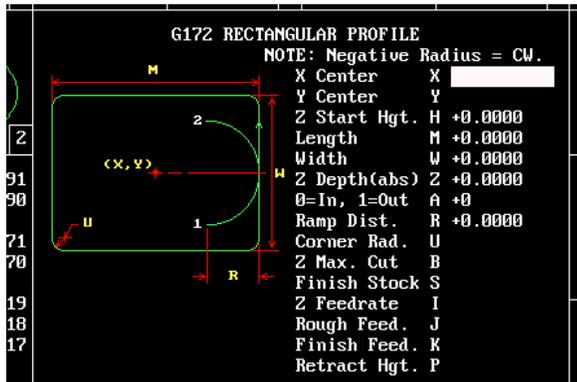
```
     G172 RECTANGULAR PROFILE
     NOTE: Negative Radius = CW.
     X Center     X
     Y Center     Y
     Z Start Hgt. H +0.0000
     Length       M +0.0000
     Width        W +0.0000
     Z Depth(abs) Z +0.0000
     0=In, 1=Out  A +0
     Ramp Dist.   R +0.0000
     Corner Rad.  U
     Z Max. Cut   B
     Finish Stock S
     Z Feedrate   I
     Rough Feed.  J
     Finish Feed. K
     Retract Hgt. P
```

Center X.	X	X Center (optional).
Center Y.	Y	Y Center (optional).
Start height.	H	Start height above surface to be cut.
Length	M	Length of pocket X axis (actual).
Width..	W	Width of pocket Y axis (actual)
Z depth (absolute).	Z	Depth to be cut (absolute).
0 = inside, 1 = outside.	A	0 insde ,1 outside of circle.
Ramp Distance.	R	Size of ramp on radius.
Corner radius	U	Radius in corners.
Z Maximum cut.	B	Maximun depth in Z per pass.
Finish Stock.	S	Amount of stock left for finish cut.
Z Feedrate.	I	Feedrate in Z axis.
Rough feedrate.	J	Feedrate for roughing.
Finish Feedrate.	K	Feedrate for finishing.
Retract height.	P	High retact if higher than H value.

Thread Mill Cycle (G181)

Format: G181 Xn Yn Zn Hn Pn Dn Cn Bn Rn Sn En Jn Kn Vn

> **WARNING: The first move in this cycle is a rapid move to the center of the thread before moving the Z axis. Make sure the tool is properly located before calling up this cycle.**

Use the thread milling for cutting inside or outside threads. It will cut either Inch or MM, left or right hand, and Z movement up or down. A single tooth or multi-toothed tool may be used. Start can be at the top or bottom of the hole or boss. The tools are set, as you would normally set TLO.

Programming the Thread Mill Cycle

To program the Thread Mill Cycle:

1. In Edit mode, press **Help** (**F1**), select **PATHS** from the menu, then "**THREAD MILL**" to display the G181 Thread Mill pop-up menu (refer to **Figure 5-20**).

2. Complete the entry fields (refer to **Table 5-28, G181 Address Words**), and press **EXIT** (**F10**).

Depending on the parameters used and their values, when cutting a thread, the tool can ramp into the cut as illustrated in **Figure 5-20**. See **Table 5-28, G181 Address Words**.

THREADMILLPU

Figure 5-20, Thread Mill Pop-up Menu

Table 5-28 describes the **Thread Mill Cycle** entry fields.

Table 5-28, G181 Address Words

Address Word	Description
X	Absolute X coordinate of the center of the thread. If no coordinate is entered, the CNC puts the center of thread at the current tool position. (Optional)
Y	Absolute Y coordinate of the center of the thread. If no coordinate is entered, the CNC puts the center of thread at the current tool position. (Optional)
Z	Absolute Z position where the thread cut will finish. This can be above or below the start position depending on the direction of the thread cut: up or down. (Required)
H	Absolute Z position where the thread cut starts. This can be above or below the finish position depending on the direction of the thread cut, up or down. If not set, cycle will use the current Z tool position. (Optional)
P	An Absolute safe Z position above the part for rapid moves in X and/or Y. (Required) **Warning: P must be above the part to avoid a crash while positioning.**
D	Major thread Diameter. If this is a tapered thread, it is the major diameter at the Z start position. Hence, if you have a tapered hole and you start at the top and cut down, you would have a different major diameter than if you started at the bottom and cut up. A plus (+) value cuts in the CW direction and a minus (-) value cuts in the CCW direction. (Required)
C	Depth of thread. The incremental depth of thread on one side. A plus (+) value is inside thread, a minus (-) value is outside thread. (Required)
B	Threads per inch (TPI) or lead of thread in MM. Note: The minimum number of threads per inch is "1". (Required)
R	Size of radius arcing into start of thread. (Optional) **Note:** If **R** is a positive value or not set and the thread is "inside", the cycle will always return to the center between passes. If **R** is a negative value, the cutter will move to the start or end point that is closest to the center if inside thread, and farthest away from center if outside thread. If **R** is not specified at all and the thread is outside, the cutter will back away from the largest diameter by an amount equal to the thread depth.
S	Amount to leave for a finish pass after the roughing passes. (Optional)
E	Number of roughing cuts to be taken. (Optional) **Note:** If "Stock" is not set or set to zero and **E** is 1 or 0, the cycle will make just one pass at the full depth. If "Stock" is set to greater than zero and **E** is 1 or 0, the cycle will make one pass at the stock depth and one pass at full thread depth. **Note:** If you would like all non-cutting positioning moves to be rapid, set **E** to a negative number.

(Continued…)

Table 5-28, G181 Address Words

Address Word	Description
J	Feedrate for roughing. (If not set (blank), the cycle will use the current active feedrate) (Optional)
K	Feedrate for the finish pass. (If not set (blank), the cycle will use the current active feedrate) (Optional)
V	Angle on one side of the thread, (not the included angle). The angle is measured from the right side going counter clockwise with a positive number and clockwise with a negative number. A standard pipe tape with an inside cut would be -1.7833. If not set (blank), than the thread is straight. (Optional)

Tool Length Offset is set the same as with any other tool or operation. A tool diameter also has to be set in the tool table, as cutter compensation is built into this cycle (cutter compensation is not allowed during the use of this cycle).

If X and Y are not programmed, position tool center of the thread before the G181 line:

- X and Y will rapid to the starting position of the thread.
- Z will rapid to the safe height specified in "P".
- The Z axis will feed down to the start cut position "H". This could be above or below the Z position specified in the "Z" finish position.
- Depending on what is in the "R" parameter the tool will arc into the first cut position.
- Spiral up or down, depending on the difference between "Z" and "H" and go counterclockwise or clockwise depending if "D" is plus or minus.
- Then arc-out and feed to the thread center for inside threads or a safe distance away from the thread for outside threads depending on the value in "R".
- Then feed back to the "H" height.
- Then feed X and Y to the next depth of cut. The depth of each roughing pass will be the thread depth specified in the "C" parameter minus the stock amount specified in the "S" parameter, divided by the number of roughing passes specified in the "E" parameter.
- The cycle repeats this process until the final finish pass.
- It will then cut the thread at the full thread major diameter.

When cutting a taper on an inside thread, care should be taken. An error will be generated if the diameter on the small end of the taper becomes too small for the tool to fit along with arc in and out moves. Not entering and arc-in value in the "R" parameter will allow the cycle to move to the center of the hole for maximum clearance.

Sample Thread Milling Cycle Program

1	G0 G90 G70 G17
2	T1 M6
3	S2000 M3
4	X0 Y0
5	G181 Z-1. H0.1 P.5 D1. C.0625 B8. R.1 S.002 E2 J20.0 K5.0
6	Z5
7	M5
8	M2

With a cutter diameter of 0.625, this program will cut a 1-8 inside thread at X0 Y0. The tool will spiral down the thread pitch of 8 threads per inch, finishing at a depth of –1. The starting height is 0.1, the safe rapid Z height is 0.5, the major thread diameter is 1 inch, and depth of thread is 0.0625. The arc-in radius is 0.1 and the stock amount for the finish pass is 0.002. The rough feedrate is 20.0 and the finish feedrate is 5.0.

Note:	If you would like all non-cutting positioning moves to be rapid, set "E" to a negative number. The idea is to initially set "E" as a positive number and after proving out the program, change "E" to a negative number for faster production. If you only need one pass to size and you want the positioning moves to be rapid, set "E" to -1.

Drilling Cycles G80 Series

COUNTERBORE G82 — 3	**PECKING** G83 — 4	**TAPPING** G84 — 5	**BORE/BI** G85 — 6	**BORE/UNI** G86 — 7	**CHIP BREAK** G87 — 8

DRILLING	FLAT BORE
G81 — 2	G89 — 9

```
DRILLING                    If they are not active, from group 1 Plane,     FLAT BORE
                            inch/metric, and absolute/incremental can
                            be selected.

G81            2            Move the cursor around the screen with the        G89        9
                            arrow keys or by number required.
Incr.      G91              Press SELECT key to pick the item. Combinations  BOLT CIRCLE
Absolute   G90              of items are not allowed in the same block and if
                            selected, an error message will be displayed.
MM         G71              Press ACCEPT to insert data into the program and
Inch       G70              remain in Help, press EXIT to insert data into
                            the program and return to the Editor.            G79        +/-
YZ Plane   G19              All of the G80 series cycles are modal and will
XZ Plane   G18              activate the Z axis at each X,Y coordinate until  PATTERN
XY Plane   G17              canceled with a G80 code.
                            NOTE: Must program drill cycle before bolt circle.
CANCEL G80  1                                                                G179        .
```

| ↑ | ↓ | | Text | Select | | Abort | Accept | Prev | Exit |

Note: That there is now a **G80** in the first box on left.

All **G80** cycles must be turn OFF with a **G80** as soon as drill operation is finished.

Basic Drilling Cycles G81
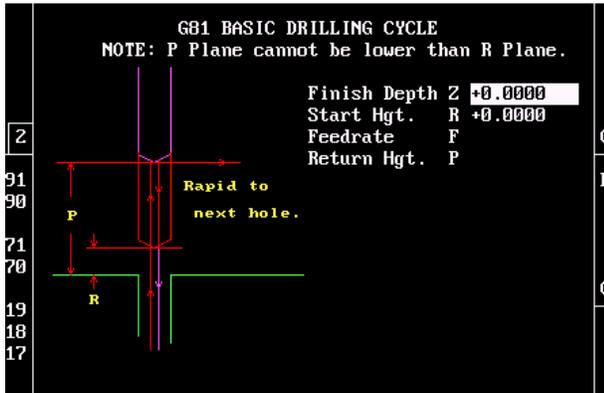
Finish Depth.   **Z**   Finish depth of hole.
Start Hgt.   **R**   Start height above surface to be drilled.
Feedrate.   **F**   Feedrate for drilling.
Return Hgt.   **P**   Return height if higher than R plane.

Counter Boring Cycles G82
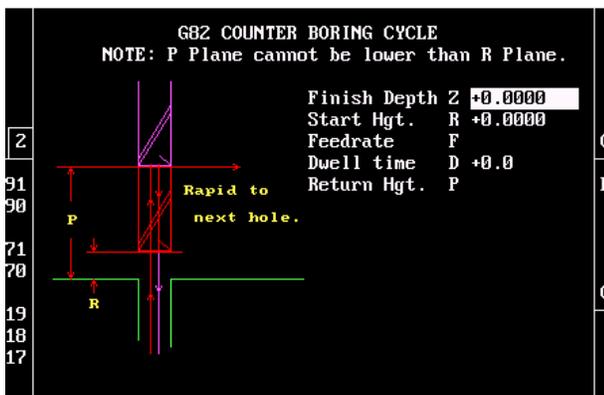
Finish Depth.   **Z**   Finish depth of hole.
Start Hgt.   **R**   Start height above surface to be drilled.
Feedrate.   **F**   Feedrate for drilling.
Dwell time   **D**   Dwell time qat bottom of hole.
Return Hgt.   **P**   Return height if higher than R plane.

Peck Drilling Cycles G83
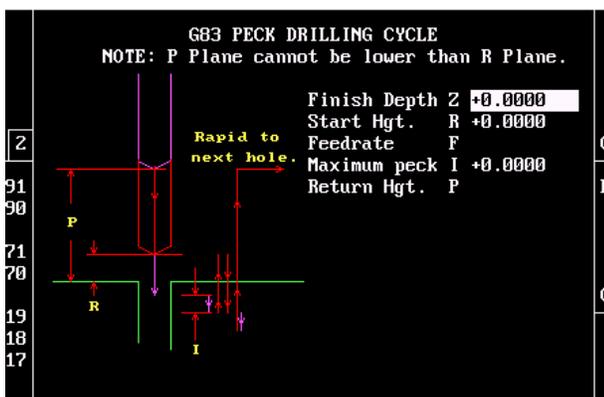
Finish Depth.   **Z**   Finish depth of hole.
Start Hgt.   **R**   Start height .1 above surface to be drilled.
Feedrate.   **F**   Feedrate for drilling.
Maximun Peck   **I**   Maximun peck before retracting.
Return Hgt.   **P**   Return height if higher than R plane.

Tapping Cycles G84

```
         G84 TAPPING CYCLE
NOTE: P Plane cannot be lower than R Plane.
        S: 0 = Spindle sync. OFF, 1 = ON.
        Program TPI in Inch or Lead in MM.
        Spindle RPM must be programmed.


          Finish Depth Z +0.0000
  Rapid to   Start Hgt.   R +0.0000
next hole.   TPI/Lead     F +0.0000
             Spindle      S
             Return Hgt.  P
             Dwell time   D
```

Finish Depth.	**Z**	Finish depth of hole.
Start Hgt.	**R**	Start height above surface to be drilled.
TPI/Lead.	**F**	TPI if inch/Lead if MM.
Spindle.	**S**	Spindle sync. 0 = OFF, 1 = ON
Return Hgt.	**P**	Return height if higher than R plane.
Dwell time	**D**	Dwell at bottom if necessary.

Boring Cycles G85

```
         G85 BORING CYCLE
NOTE: P Plane cannot be lower than R Plane.

          Finish Depth Z +0.0000
          Start Hgt.   R +0.0000
          Feedrate     F
          Return Hgt.  P
  Rapid to
next hole.
```

Finish Depth.	**Z**	Finish depth of hole.
Start Hgt.	**R**	Start height above surface to be drilled.
Feedrate.	**F**	Feedrate for drilling.
Return Hgt.	**P**	Return height if higher than R plane.

Boring Cycles One Direction G86

```
      G86 BORING IN ONE DIRECTION
NOTE: P Plane cannot be lower than R Plane.

          Finish Depth Z +0.0000
          Start Hgt.   R +0.0000
          Feedrate     F
          X Backoff    I
  Rapid to   Dwell time   D
next hole.   Return Hgt.  P
          Index Angle  C
```

Finish Depth.	**Z**	Finish depth of hole.
Start Hgt.	**R**	Start height above surface to be drilled.
Feedrate.	**F**	Feedrate for drilling.
X Backoff.	**I**	Backoff before retracting from hole.
Dwell time.	**D**	Dwell to flat bottom hole.
Return Hgt.	**P**	Return height if higher than R plane.
Index Angle.	**C**	Index angle to orient spindle to backoff.

Chip Breaking Cycle G87
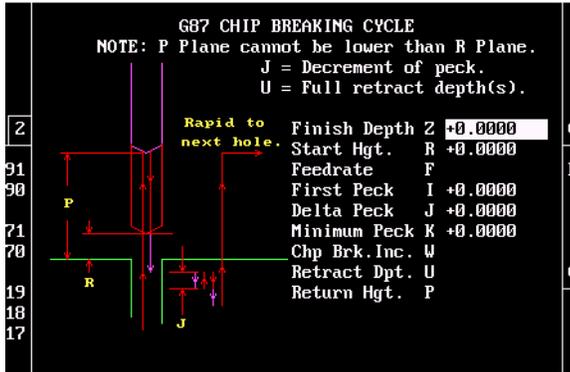
```
G87 CHIP BREAKING CYCLE
NOTE: P Plane cannot be lower than R Plane.
              J = Decrement of peck.
              U = Full retract depth(s).

Rapid to    Finish Depth Z +0.0000
next hole.  Start Hgt.   R +0.0000
            Feedrate     F
            First Peck   I +0.0000
            Delta Peck   J +0.0000
            Minimum Peck K +0.0000
            Chp Brk.Inc. W
            Retract Dpt. U
            Return Hgt.  P
```

Finish Depth.	**Z**	Finish depth of hole.
Start Hgt.	**R**	Start height .1 above surface to be drilled.
Feedrate.	**F**	Feedrate for drilling.
First Peck.	**I**	Amount of first peck.
Delta Peck.	**J**	Amount to decrease peck each peck.
Minimum Peck.	**K**	Smallest peck amount.
Chp. Brk. Inc.	**W**	Retract for chip break
Retract Depth.	**U**	Depth full retract accures.
Return Hgt.	**P**	Return height if higher than R plane.

Flat Bottom Boring Cycle G89
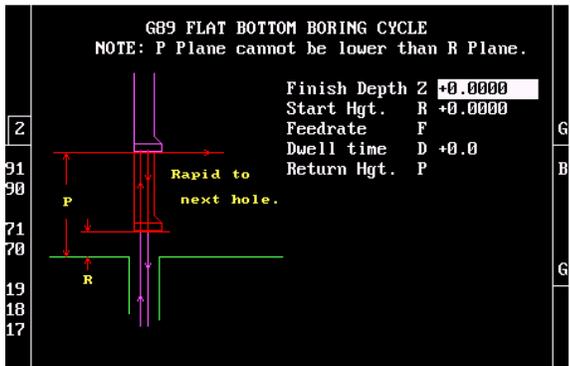
```
G89 FLAT BOTTOM BORING CYCLE
NOTE: P Plane cannot be lower than R Plane.

            Finish Depth Z +0.0000
            Start Hgt.   R +0.0000
            Feedrate     F
            Dwell time   D +0.0
            Return Hgt.  P
Rapid to
next hole.
```

Finish Depth.	**Z**	Finish depth of hole.
Start Hgt.	**R**	Start height above surface to be drilled.
Feedrate.	**F**	Feedrate for drilling.
Dwell.	**D**	Dwell in second at bottom of hole.
Return Hgt.	**P**	Return height if higher than R plane.

Bolt Hole Circle Drilling G79

```
        G79 BOLT HOLE CIRCLE DRILLING
    NOTE: You must program G81 - G89 before G79.

                    Center      X  [      ]
                    Center      Y
                    Index Angle C
              A     First Angle A  +0.00000
                  C*Last Angle  B
                    Number holes H +0
              B  Deg.Diameter   D  +0.0000
                 0  Radial Path R

            * C-Optional shifted angle.
```
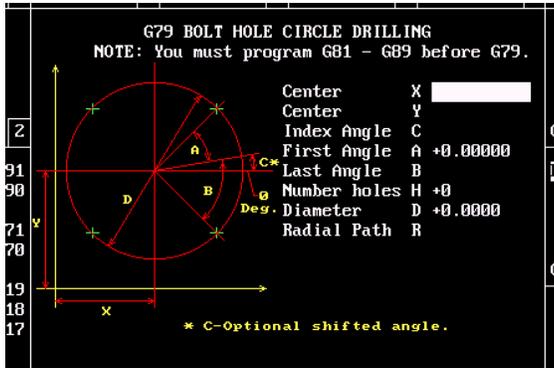
Center.	**X**	Center X axis.
Center.	**Y**	Center Y axis.
Index Angle.	**C**	Angle to rotate 0 angle from 3 o'cclock.
First Angle.	**A**	Angle of first hole from 0.
Last Angle.	**B**	Angle of last hole, if full pattern not required.
Number Holes.	**H**	Number of holes to drill.
Diameter.	**D**	Diameter of pattern.
Radial Path.	**R**	If 1 is entered will move radially around pattern

Hole Pattern Dilling G179

```
        G179 HOLE PATTERN DRILLING
    NOTE: You must program G81 - G89 before G179.

                    X Start     X  +0.0000
                    Y Start     Y  +0.0000
         +  +  +  + Angle       C
              V     X Length    A
         +  +  +  + Num. X holes B +0
                    Y Width     D
         +  +  +  + Num. Y holes E +0
           U        X Increment U
                    Y Increment V
                    Pat.0 Sqr.1 W +0
         X     A
```

X Start.	**X**	Start point X axis.
Y Start.	**Y**	Start point Y axis.
Angle.	**C**	Angle If pattern is rotated.
X Length.	**A**	Distance from first to last hole X axis.
Y Width	**B**	Distance from first to last hole Y axis.
Num. Holes X.	**D**	Number of holes X axis.
Num. Holes Y.	**E**	Number of holes Y axis.
X Increment.	**U**	Distance between holes X axis.
Y Increment.	**V**	Distance between holes Y axis.
Pat.=0 Sqr.=1	**W**	Pattern as shown or square around outside.

Use D & E or U & W not both.

G-Code without Graphics

```
        MISCELLANEOUS G CODES
G04 Dwell.
G09 Exact Stop (Single Block).
G22 Stroke Limit.
G28 Reference Point Return.
G29 Return from Reference Point.
G53 Fixture Offset(s) (Coord. Syst. Select)
G61 Exact Stop Mode (Contouring Mode OFF).
G64 Contouring Mode (Exact Stop Mode OFF).
G65 Macro Call, Single (Non-Modal)
G66 Macro Call, Modal.
G67 Cancel Modal Macro.
G92 Preset Zero.
G94 Per Minute Feed
G95 Per Revolution Feed.
```

Dwell In seconds with **T**n
Exact stop will stop exactly in positiion one shot.
Stroke Limit set a box that tool cannot move outside or inside.
Reference Point Return sends machine home in designated axis.
Reference Point Return
Fixture offset Absolute zero shift from Home.
Exact Stop will stop after each move to get into exact position.
Contouring Mode continuous path no stops between moves.
Macro Call one shot user writen macro.
Macro Call modal macro has to be turned off.
Cancel Macro cancel modal macro.
Preset Zero increamental zero shift canceled by G53 or homing.
Feed Per Minute feed in inches per minute.
Feed Per Revolution feed in inches per revolution.

Miscellaneous M-Codes

```
            M-CODES
M0 Program Stop.
M2 End of Program
M3 Spindle ON FWD.
M4 Spindle ON REV.
M5 Spindle OFF.
M8 Coolant ON.
M9 Coolant OFF.
M30 Jump to New Program
M98 Call Subprogram.
M99 End Subprogram.
M100 Mirror Image.
M105 Dry-Run, All Axes.
M106 Dry-Run, NO Z Axis.
M107 Dry-Run OFF (Cancel M105,M106)
```

Program Stop stop program until START is pressed to to continue.
End of Program end of main program.
Spindle ON Forward turn spindle ON FORWARD.
Spindle ON Reverse turns spindle ON REVERSE.
Spindle OFF turns spindle OFF.
Coolant ON tuns coolant ON.
Coolant OFF turns coolant OFF.
Jump to New Program entered using **P**xxxx.
Call Subroutine call a subroutine using **P**xx.
End Subroutine Last line of subroutine.
Mirror Image axis need to be entered X,Y or Z.M100 turns OFF
Dry Run All Axis display shows motion but no table movement.
Dry Run NO Z Axis X and Y move no Z axis movement.
Dry Run OFF turns OFF dry run.

5000M CNC Control

Program Management

INTRODUCTION

The Program Directory provides access to all the program management and disk utilities. These functions include Creating, Selecting, Deleting, Undeleting and Copying programs.
The Program Directory also provides access to the Floppy Drive utilities.

Accessing PROGRAM DIRECTORY page.

(1) From the MANUAL mode press: **Program**   The Program files are listed in alphabetical order.

To access a PROGRAM file from the Program page.

There are 2 methods to access a particular Program File:

(1) Using the ⊗ keys move the High Light to the desired program and Press: **Edit**

or

(2) Press the first letter of the desired program name, this will move the High Light to the first program

name with that letter, then use the ⊗ keys to move the High Light to the desired program

and Press: **Edit**

To CREATE a new Program.

NOTE: There are 2 methods of creating a program file, both are shown below.

Method No.1 Creating a new program file. From the PROGRAM page:

(1) Press: **Create**   At the prompt `NEW PROGRAM: _` type in the new program name

`NEW PROGRAM: EXAMPLE_` and Press: ⬊

NOTE: Program names can be up to 8 characters in length, but may not include spaces or periods.

Method No.2 Copying and using an existing program.
If a similar program to that required already exists, it may be copied and given a new name.
This allows similarly formatted programs to be used without having to re-type the information.

(1) Using the [arrow keys] keys High Light the existing program to be copied [SAMPLE-1.G / SAMPLE-2.G]

Press: **Utility** this brings up a menu, [►Copy / Rename] with Copy High Lighted

Press [arrow key] This will display a second menu [►Other...] High light Other and Press [arrow key]

This will bring up a third menu [► Copy to: EXAMPLE_] type in the new program name

"EXAMPLE" and Press [arrow key] this will create a new program named EXAMPLE and also leave the

original program "SAMPLE1.G" intact. This copied program can now be edited to suit the new part.
It is not necessary to type the file extension (.G) as this is completed automatically.

To change or EDIT an existing program.

(1) High Light the program to be changed / edited using the [arrow keys] keys.

(2) Press: **Edit** to enter the file editor .

To DELETE an existing program.

(1) High Light the program to be DELTED using the [arrow keys] keys. Press **F3** **Delete**

(2) To avoid accidental deletion of programs the system requests verification of deletion

[Delete EXAMPLE.G?] To delete the program Press **F1** **Yes**

3.

To UNDELETE / RESTORE a deleted program.

Should a program become accidentally deleted the following method can be used to restore the program.

NOTE: The restore capability of the system is determined by the space available on the drive. If the space on the drive where the program was stored has been reused restore will not be an available function.

(1) From the PROGRAM page Press:　**Utility**

(2) Using the　keys, High Light　Restore

Print
▶Restore
Floppy Format

Press

(3) The system will list any and all programs that are available to be restored. Using the keys High Light the program to be restored.

(4) Type in the first letter of the program to be restored (the first letter of the program name is lost when a program is deleted).

(5) Press　**Cont**　if the program can be restored the system will place the program file in its

alphabetical location.

NOTE: ALWAYS CHECK THE VALIDITY OF A PROGRAM AFTER RESTORING. SOME OF THE INFORMATION WITHIN THE PROGRAM MAY HAVE CHANGED.

To SELECT a program to run in the AUTO mode.

Once a program has been created and verified to be correct the operator must SELECT the program to run when in the Auto or Single Step mode.

(1) From the PROGRAM page using the　keys High light the program name.

(2) Press　**Select**　the selected program to be run will now be displayed in the lower right corner

of the screen.　Selected Program: EXAMPLE.G

4.

To COPY a program to the A: Drive (Floppy Disk).

The 5000M control uses the C:\ drive and the USER directory to store programs within the system. When in the PROGRAM directory the drive and directory are displayed in the lower left corner of the screen.   C:\USER

The 5000M uses the A:/ drive for the Floppy Disk Drive. Programs may be copied to or copied from the Floppy Drive individually or in multiples.

(1) From the PROGRAM directory use the   keys to High light the program to be copied to the

A:\ (Floppy Disk) drive. (Place a floppy disk in the A:\ floppy disk drive unit).

(2) If multiple programs are to be copied High light the first program to be copied using the keys

and Press   Then using the   keys again High light the next program to be copied and

Press   Continue until all programs to be copied are High lighted.

(3) When all programs to be copied are High light Press   **Utility**

(4) With Copy Hi-lited   ►Copy Rename   Press

(5) This brings up a menu   Copy to: ►A: C: With A: High lighted Press

The lower left corner of the screen displays COPYING…. while the program is copied. The procedure

is complete when COPYING…. is no longer displayed.

To COPY a program from the A: Drive (Floppy Disk) to the Program Directory C:\USER

The 5000M uses the A:/ drive for the Floppy Disk Drive. Programs may be copied to or copied from the Floppy Drive individually or in multiples. To view or copy the programs on a Floppy Disk, the operator must first LOG to the Floppy disk Drive (A:\)

(1) From the PROGRAM directory Press [⇧] (Shift) this will change the descriptions of the F keys,

Press [**Log**] This shows a menu of the drives that the operator can view on screen.

(2) [Log to: ▶A: C:] High light A: and Press [⇨] the programs on the Floppy Dive (A:\) will now be displayed.

(3) Use the [✛ arrow keys] keys to High light the program(s) to be copied to the Program Directory (C:\)

If multiple programs are to be copied High light the first program to be copied using the [✛ arrow keys] keys and Press [⇨] Then using the [✛ arrow keys] keys again High light the next program to be copied and Press [⇨] Continue until all programs to be copied are High lighted.

(4) When all programs to be copied are High lighted Press [**Utility**]

(5) With Copy High lighted [▶Copy Rename] Press [⇨] A menu is shown [Copy to: A: ▶C: D:]
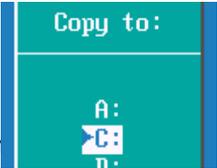
With C: High lighted Press [⇨] The lower left corner of the screen displays COPYING. while the program(s) are copied. The procedure is complete when COPYING…. is no longer displayed.

6.

Continued….

To return to the Program Directory , LOG back to C:\USER

(1) Press ⬆ (Shift) this will change the descriptions of the F keys,

(2) Press **Log** This shows a menu of the drives that the operator can view on screen.

Log to:

A:
▶C:

(2) High light C: and Press ➡ the C:\USER (lower left corner of the screen)

will now be displayed.

To COPY program files to the A: (Floppy Disk) when using the CAM

When using the CAM Editor to write programs the 5000M system will create additional program files, and ALL these files MUST BE COPIED to ensure correct operation of the program.
The program name to which the CAM is to be associated MUST be High lighted prior to entering CAM or (F4 from the Program Directory). this program will have extension of **.G**
The correct program must be high light because upon entering the CAM , the system creates a file with the same name as the program but with an extension of **.CAM** and also a file with the extension of **.GEO**
These files are used to record the CAM (.CAM file) settings inside the CAM and also to record the GEOMETRICAL ELEMENTS created (.GEO file).

Once a Shape has been created a 3rd file is also created, this file will have an extension of **.1**
NOTE: If more than one Shape is created the shape files will have extensions of **.2 .3** etc. assigned in the order in which they were created.

Example: original program   PROGRAM.G
                file created by shape editor   PROGRAM.CAM
                file created by shape editor   PROGRAM.GEO
                first shape file created   PROGRAM.1
                second shape file created   PROGRAM.2
                third shape file created   PROGRAM.3   etc…..
                first tool path   PROGRAM.T1
                second tool path   PROGRAM.T2
                third tool path   PROGRAM.T3 etc…..

7.

To COPY all the program files to the A: (Floppy Disk) when using the CAM continued…..

The 5000M uses the C:/USER drive in the Program Directory. Upon entering the Program Directory the programs displayed will all have an extension of **.G**
To view all the program files described on the previous page the operator must first LOG to the the root of C:\USER directory (this is where all program files with any extension can be viewed). This is achieved by using wildcard symbols. These symbols are *** . *** (any program name with any extension).

(1) From the PROGRAM directory Press ⬆ (Shift) this will change the descriptions of the F keys,

Press **Log**   This shows a menu of the drives that the operator can view on screen.

(2) High light Other: `Other...`   and Press ➡

(3) At the next menu `▶ Log to:  *.*_`   type ***.*** Press ⬆ *• *• ⬆ *•

(4) Press ⬇   The screen will now display all programs with all extensions.

```
..▯
EXAMPLE.1
EXAMPLE.CAM
EXAMPLE.G
EXAMPLE.GEO
SAMPLE-1.G
SAMPLE-2.G
SAMPLE-3.G
SAMPLE-4.G
SAMPLE-5.G
SAMPLE-6.G
SAMPLE-7.G
SAMPLE-8.G
SAMPLE-9.G




C:\USER                          Selected Program:

   Create  Delete  Shapes   List   Select   Draw   Edit  Utility  Exit
```
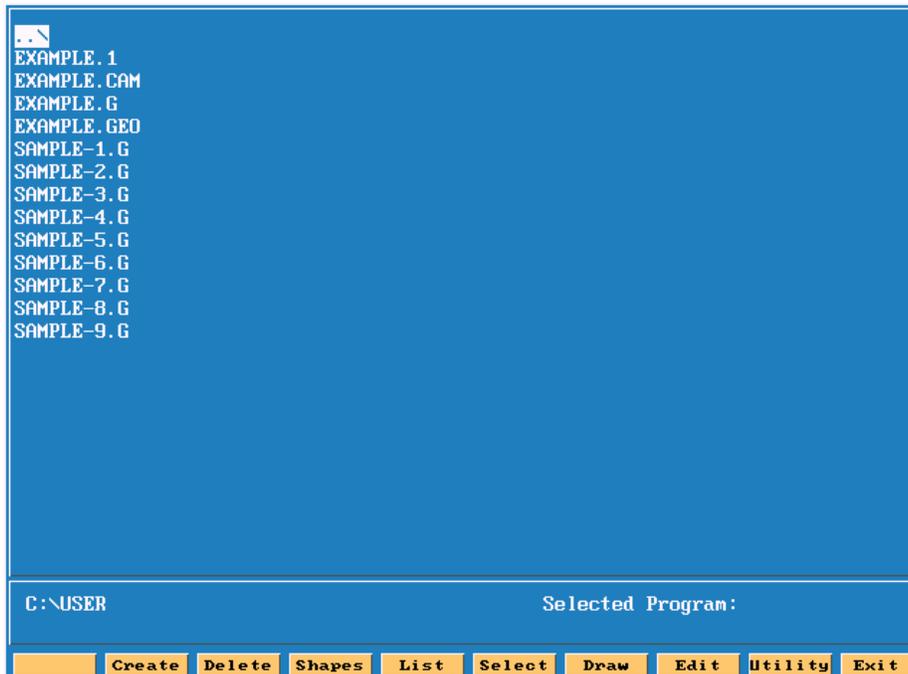
To COPY all the program files to the A: (Floppy Disk) when using the CAM continued…..

An alternative method is also provided to obtain the screen showing all programs with all extensions Directory.

This method is completed using the [⇧] and **Display** keys.

This sequence of key strikes toggles the Program Directory display through the following screens. The key strikes are repeated to toggle through the screens shown below.

```
EXAMPLE.G
SAMPLE-1.G
SAMPLE-2.G
SAMPLE-3.G
SAMPLE-4.G
SAMPLE-5.G
SAMPLE-6.G
SAMPLE-7.G
SAMPLE-8.G
SAMPLE-9.G




C:\USER

        Create
```
Screen on entry into Program Directory (displays .G programs only)

```
EXAMPLE.G          716  10/12/1999  03:36 pm
SAMPLE-1.G       1,114  02/16/1999  11:42 am
SAMPLE-2.G       1,184  02/16/1999  11:45 am
SAMPLE-3.G       1,053  02/16/1999  11:45 am
SAMPLE-4.G       1,686  02/16/1999  11:46 am
SAMPLE-5.G       1,725  02/16/1999  11:46 am
SAMPLE-6.G       2,339  02/16/1999  11:46 am
SAMPLE-7.G       1,566  02/16/1999  11:46 am
SAMPLE-8.G       2,817  02/16/1999  11:47 am
SAMPLE-9.G       3,557  02/16/1999  11:47 am
```
Screen after 1st sequence (displays .G programs and size date and time created)

```
..\
EXAMPLE.1
EXAMPLE.CAM
EXAMPLE.G
EXAMPLE.GEO
SAMPLE-1.G
SAMPLE-2.G
SAMPLE-3.G
SAMPLE-4.G
SAMPLE-5.G
SAMPLE-6.G
SAMPLE-7.G
SAMPLE-8.G
SAMPLE-9.G

C:\USE

    Create
```
Screen after 2nd sequence (displays all programs *.*)

```
..\                <DIR>  04/19/1999  03:18 pm
EXAMPLE.1            594  11/19/1999  02:05 pm
EXAMPLE.CAM       6,820  11/19/1999  02:05 pm
EXAMPLE.G           716  10/12/1999  03:36 pm
EXAMPLE.GEO       3,302  10/14/1999  04:41 pm
SAMPLE-1.G        1,114  02/16/1999  11:42 am
SAMPLE-2.G        1,184  02/16/1999  11:45 am
SAMPLE-3.G        1,053  02/16/1999  11:45 am
SAMPLE-4.G        1,686  02/16/1999  11:46 am
SAMPLE-5.G        1,725  02/16/1999  11:46 am
SAMPLE-6.G        2,339  02/16/1999  11:46 am
SAMPLE-7.G        1,566  02/16/1999  11:46 am
SAMPLE-8.G        2,817  02/16/1999  11:47 am
SAMPLE-9.G        3,557  02/16/1999  11:47 am

C:\USER

        Create
```

Screen after 3rd sequence (displays all programs *.* and size, date and time created)

```
EXAMPLE.G
SAMPLE-1.G
SAMPLE-2.G
SAMPLE-3.G
SAMPLE-4.G
SAMPLE-5.G
SAMPLE-6.G
SAMPLE-7.G
SAMPLE-8.G
SAMPLE-9.G

C:\USER

    Crea




C:\USER                        Selected Program:

    Create  Delete  Shapes  List  Select  Draw  Edit  Utility  Exit
```

Screen after 4th sequence returns to the Program Directory (displays .G programs only)

To COPY all the program files to the A: (Floppy Disk) when using the CAM continued…..

To COPY all the EXAMPLE program files shown below complete the following:

(1)Use the [arrow keys icon] keys to High light the first program to be copied (EXAMPLE.1) to the Floppy Drive

(2) Using the [arrow key icon] key High light the remaining EXAMPLE program files
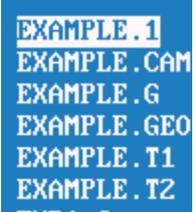
```
EXAMPLE.1
EXAMPLE.CAM
EXAMPLE.G
EXAMPLE.GEO
EXAMPLE.T1
EXAMPLE.T2
```

(3) Press **Utility** and with Copy Hi-lited [Copy / Rename menu] Press [arrow key icon]

a menu is displayed [Copy to: A: C: menu] with A: Hi-lited Press [arrow key icon] The lower left corner of the

screen displays **COPY 6 MARKED FILES ?** Press [arrow key icon]

COPYING…. will be displayed while the programs are copied. The procedure is complete when

COPYING…. is no longer displayed.

To COPY program files (all extensions) from the A: (Floppy Disk) to the C:\USER directory.

(1) From the Program Directory (C:\USER) use either method described previously to display the programs with all the extensions.

(2) Insert the Floppy Disk into the Drive

(3) LOG to the A: Drive

(4) High light the program files to Copy to C:\USER   Press ENTER

(5) LOG back to the C: Drive

5000M CNC CONTROL
Simple Program

CREATING A PROGRAM

From Manual page press | **F2** | **PROGRAM**

The softkeys will changed as shown below.

| | Create | Delete | CAM | List | Select | Draw | Edit | Utility | Exit |

Press | **F2** | **CREATE**

Type the program name (Maximum 8 letters/numbers).

Press Enter put into program directory

```
LAWRENCE.G     NICE2.G        SHUTTLE.G      TNC.G
LETR230.G      07_SAMPL.G     SLOTMAC.G      WINDOW-M.G
LETRCAL1.G     PAGODA2.G      SNOWMAN.1      WOMAN.G
LETSAMBL.G     PHIL.G         SNOWMAN.CAM    WY.G
LTRS230.G      POCK.G         SNOWMAN.G
MAINTEST.G     POCK.M         SNOWMAN.GEO
MATT1008.G     POCKETS.G      SNOWMAN.T1
MER-FIN.G      POCKETS.M      SNOWMAN.T2
MER-ROU.G      POCKETS1.G     SPIRAL.G
MERC-ARC.G     POCKETS1.M     SPIRAL.S
MERC-LIN.G     PROGRAM.G      STOCKVAR.G
MERC-LIN.M     REPEAT.M       T.G
MERC1.G        RICH.G         T.NC
MERC2.G        RON.G          TAPE.G
MERC3.EXE      ROUGH.G        TAPE.S
MERC3.G        ROUGH1.G       TAPE3200.G
MILL.G         ROUGH22.G      TAPE3200.M
MILL.M         SAM.G          TAPE3200.NCC
MILL1.G        SAM1.G         TAPPING.G
MODERN.G       SAM2.G         TC.G
MONARCH.G      SCHWADA1.G     TEST.G
NATHAN.G       SCHWEDA.G      TEST2.G
NATHAN.M       SCHWEDA4.G     THREAD.G
NICE.G         SHOW98HA.G     TIM.G

C:\USER                            SELECTED PROGRAM:

      Create  Delete   CAM    List   Select   Draw    Edit  Utility  Exit
```

High light will be on program just created.

Press **F8**   **EDIT**

2.

This is how screen will appear when entering **EDIT**


```
Program: PROGRAM.G                                          1:1
  Help    Del    Ins    DelBlk   PgUp    PgDn   Move  Editing  Misc   Exit
```

Program name

```
  Help    Del    Ins    DelBlk   PgUp    PgDn   Move  Editing  Misc   Exit
```

**Note:** Insert is turn ON ,if not will not go past bottom of page.

Note: Softkeys have change .

X0 Y0

1.5

.5

.75

2.75
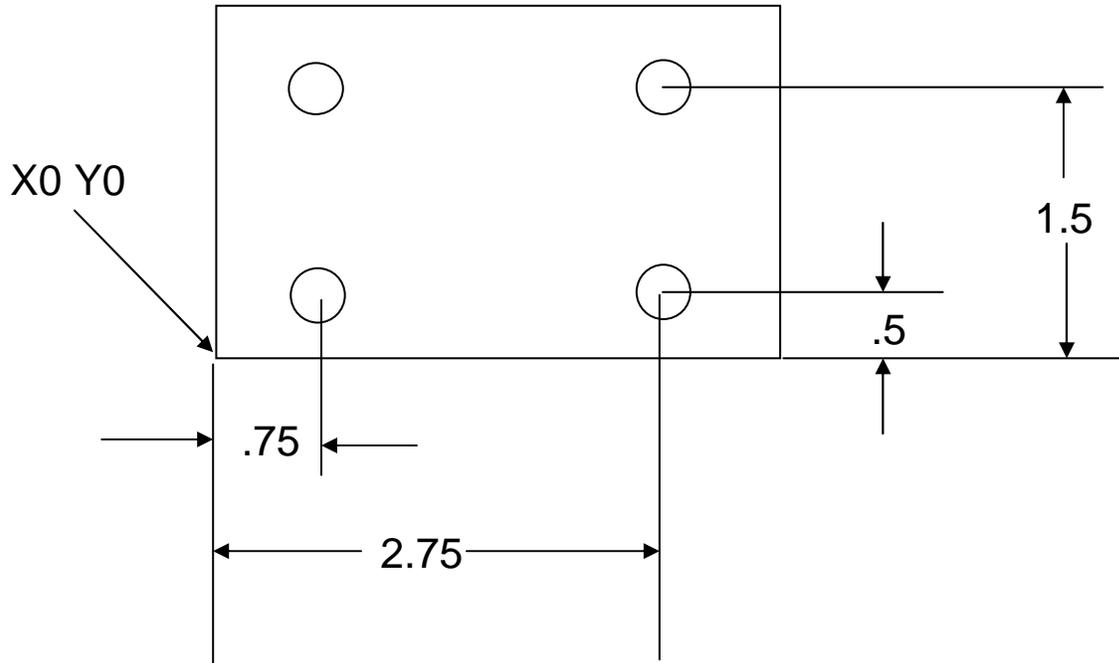
This part needs to have 4 hole drilled .25 dia and .5 deep

Press **F1**   **HELP**
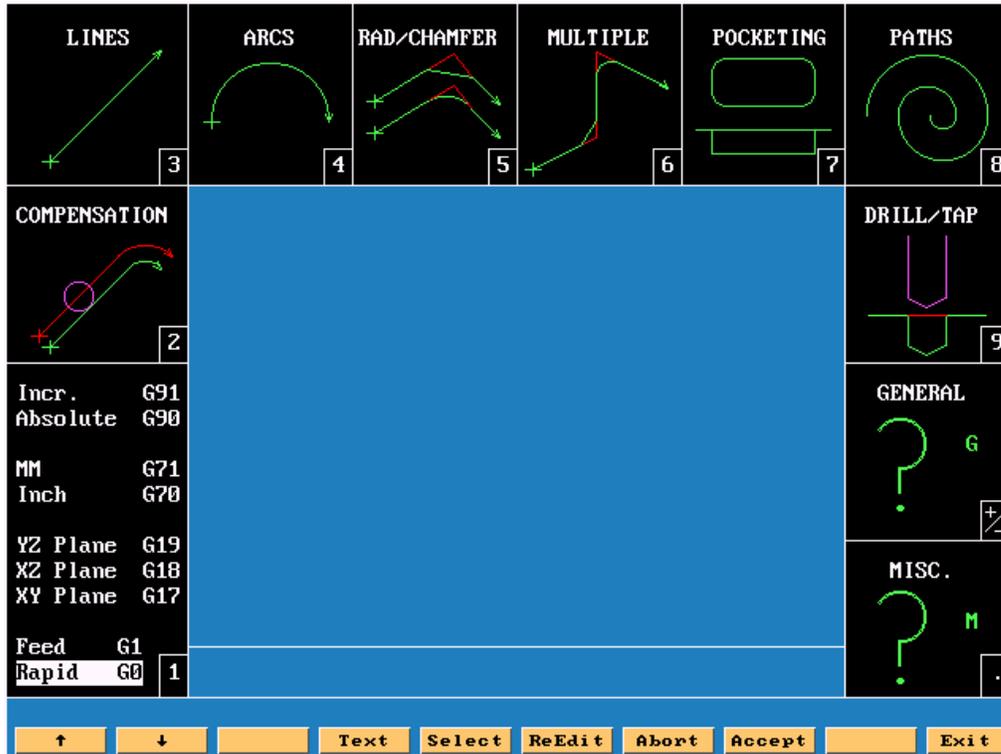
High light will be on **G0** press enter

Move high light **G17 , G70** and **G90** press enter on each of these.

Press **M5** press

Press **F10**   **Exit**   it will go into program as shown below.

G00 G17 G70 G90 M5

Top line of program.

```
G00 G17 G70 G90 M5        *RAPID,XY PLANE,INCH,ABSOLUTE,POSITION,SPINDLE OFF

G00 G17 G70 G90 M5        *RAPID,XY PLANE,INCH,ABSOLUTE,POSITION,SPINDLE OFF
T1 M6                     *LOAD TOOL,M6 ONLY REQUIRED IF TOOL CHANGER INSTALLED
S2000 M3                  *SET SPINDLE SPEEDAND TURN ON SPINDLE
_




Program: PROGRAM.G (edited)                                              4:1
```

| Help | Del | Ins | DelBlk | PgUp | PgDn | Move | Editing | Misc | Exit |

Program has been edited

Cursor position line and column

Lines 2 and 3 are typed in manually.
The * allows commence into program control will read
ignore after it.

Press | **F1** | HELP | press #9 for **DRILL** press

Screen will appear as shown below.

Press #2 for basic drilling, press enter.

```
              G81 BASIC DRILLING CYCLE
    NOTE: P Plane cannot be lower than R Plane.

                        Finish Depth  Z  -0.5000
                        Start Hgt.    R  +0.1000
                        Feedrate      F  +12.0
                        Return Hgt.   P
              Rapid to
              next hole.
    P

      R
```
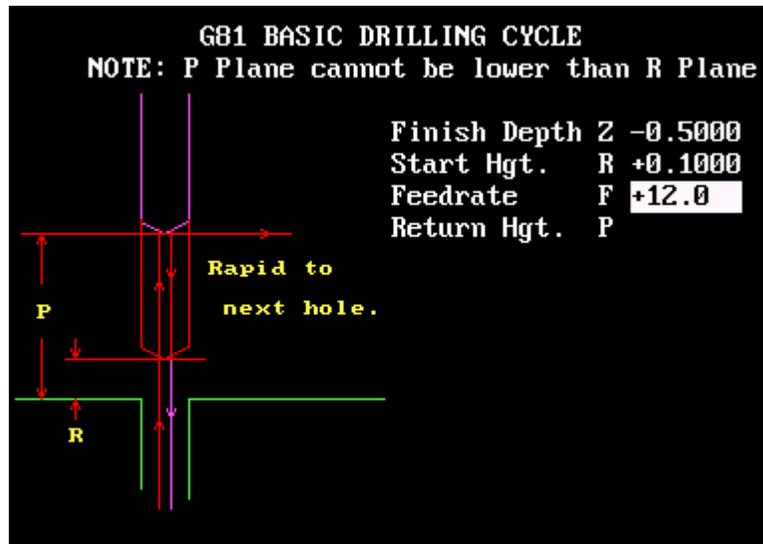
Enter values <u>Finish Depth</u> -.5 press down arrow, enter
<u>Start Hgt</u> .1 down arrow and <u>Feedrate</u> 12

Press | **F10** | **Exit** | exits help menu and enters line into program.

```
G00 G17 G70 G90 M5      *RAPID,XY PLANE,INCH,ABSOLUTE,POSITION,SPINDLE OFF

G00 G17 G70 G90 M5      *RAPID,XY PLANE,INCH,ABSOLUTE,POSITION,SPINDLE OFF
T1 M6                   *LOAD TOOL,M6 ONLY REQUIRED IF TOOL CHANGER INSTALLED
S2000 M3                *SET SPINDLE SPEEDAND TURN ON SPINDLE
G81 Z-0.5000 R0.1000 F12.0 *BASIC DRILLING CYCLE
```

The next enter hole positions.

Type **G0 X.75 Y.5** → **X2.75** → **Y1.5** → **X.75** →

Type **G80** →

Cancel drill, this must always in program after last hole.

```
G00 G17 G70 G90 M5        *RAPID,XY PLANE,INCH,ABSOLUTE,POSITION,SPINDLE OFF

G00 G17 G70 G90 M5        *RAPID,XY PLANE,INCH,ABSOLUTE,POSITION,SPINDLE OFF
T1 M6                     *LOAD TOOL,M6 ONLY REQUIRED IF TOOL CHANGER INSTALLED
S2000 M3                  *SET SPINDLE SPEEDAND TURN ON SPINDLE
G81 Z-0.5000 R0.1000 F12.0
X.75 Y.5                  *FIRST HOLE LOCATION
X2.5                      *NEXT LOCATION
Y2                        *NEXT LOCATION
X.75                      *NEXT LOCATION
G80                       *CANCEL DRILL
Z5                        *RAISE Z AXIS TO ACLEARANCE POSITION
X-2 Y4                    *MOVE X AND Y TO ALLOW ACCESS TO PART
M2                        *END OF PROGRAM_
```

Type in last three lines.

Press **F10** | Exit | press **F7** | Draw

Soft key will change as shown below.

| Edit | Run | VIEW | DISPLAY | | | | Parms | Exit |

Edit program

Run program

Fit
Window
Half
Double
Scale
Pan
Erase

Fit to screen
Zoom window
Half current size
Double current size
Scale drawing
Pans part around screen
Erase screen

Exit draw mode

View **XY plane**
View **XZ plane**
View **YZ plane**
View **isometric**

XY
XZ
YZ
Iso

Displays text ---------------- ▸Text....... On
Displays tool------------------ Tool....... On
Show tool comp------------- ToolComp... Both
Rapid ON/OFF-------------- Rapid...... On
Displays axis---------------- Axes....... On
Grid ON/OFF---------------- Grid....... None
Size of grid------------------- Grid size.. 1.0000
Auto/single step/motion--- Mode....... Auto
Run ON/OFF---------------- Run........ On
Erase screen---------------- Erase...... On
Block to start draw--------- Start N#... Start
Block to end draw---------- End N#..... End

Press **F5** | DISPLAY | cursor will be on **FIT** press enter

This will fits drawing to screen.


```
Blk: 12
X    -2.0000
Y     4.0000
Z     5.0000
F: 200.0
T:    1
D:    0.2900
L: -10.0000
Loop:     0
Rapid
Abs    Inch
Off    XY



View: XY
Comp: Both
Mode: Auto

Start Thru
End
PROGRAM.G
X-2 Y4          *MOVE X AND Y TO ALLOW ACCESS TO PART
M2              *END OF PROGRAM
Program ended normally.
Program has no comp.
        Edit    Run    VIEW    DISPLAY              Parms   Exit
```
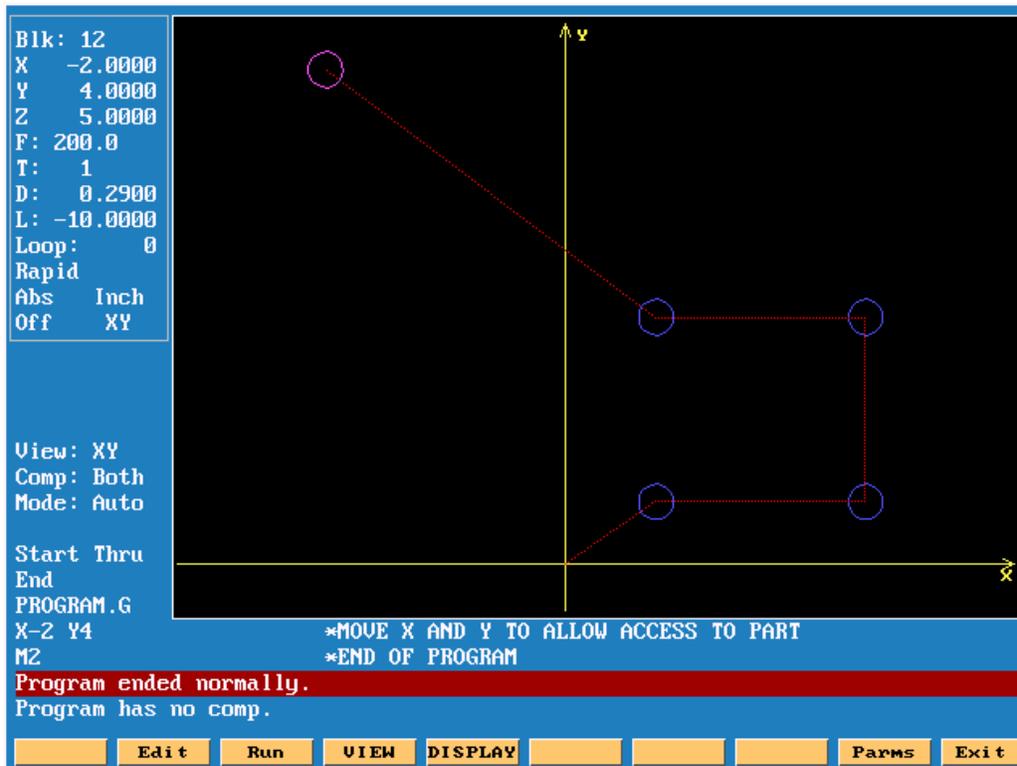
Red lines are **RAPID** moves , blue circle are the holes and purple circle is the tool.

Press **F10** | Exit | this brings control back to program page

Press **F6** | Select | to select program ,check that high light is on correct program.

Press **F10** | Exit | go to **Manual**

SETTING PART ZERO.

If using an edge finder , touch edge of part using jog keys.

Go to tool page press **F9** [Tool] press **F1** [OFFSETS]

```
                        G53 Offset                          AT

              X          Y          Z          U          W
      1. ▶ -10.0000   -10.0000    0.0000     0.0000     0.0000
      2.   -20.0000   -10.0000    0.0000     0.0000     0.0000
      3.     0.0000     0.0000    0.0000     0.0000     0.0000
      4.     0.0000     0.0000    0.0000     0.0000     0.0000
      5.     0.0000     0.0000    0.0000     0.0000     0.0000
      6.     0.0000     0.0000    0.0000     0.0000     0.0000
      7.     0.0000     0.0000    0.0000     0.0000     0.0000
      8.     0.0000     0.0000    0.0000     0.0000     0.0000
      9.     0.0000     0.0000    0.0000     0.0000     0.0000
     10.     0.0000     0.0000    0.0000     0.0000     0.0000
     11.     0.0000     0.0000    0.0000     0.0000     0.0000
     12.     0.0000     0.0000    0.0000     0.0000     0.0000
     13.     0.0000     0.0000    0.0000     0.0000     0.0000    AT

          CalibX  CalibY  CalibZ  CalibU  CalibW  Exit
```

Cursor to required **OFFSET** using arrow up and down keys.
Move X axis until part is located with edge finder.

Press **F4** [CalibX] is will enter value into table.

Because the edge find is .2 dia the position will be incorrect , press
the letter **A** key ,a box as shown below will appear.
Type in **X.1** this will adjust offset to correct position press **ENTER**.

```
▶Enter axis and adjustment value:     X.1_
```

Repeat for **Y** axis

All offset are taken from machine Home position.

12.

SETTING TOOL LENGTH OFFSETS

Put tool in spindle , bring tool tool down to top of part.

Press **F9**   **Tool**   high light correct tool number in tool page

press **F8**   **CalibZ**

If there are more move Z axis up change to next tool
and calibrate it as before,tool #0 must always be active
when setting offsets.

Press **F10**   **Exit**   when all tools are calibrated.

RUNNING PART.

Press **F6**   **Auto**   turn down feedrate override knob to 10%

% FEED
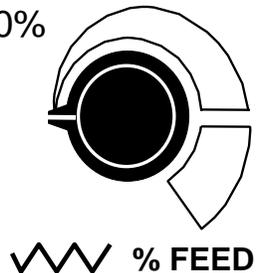
Press
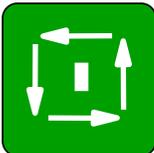
If machine has a tool changer ,it will put tool #1in spindle if
not already there.
If no tool changer install it will stop and allow you to change
tool if necessary .

When program complete **Z** axis will go up to 5" above part.

5000M CNC CONTROL
GENERATING PROGRAMS USING CAM

ANILAM

STEP BY STEP CREATION OF A PROGRAM USING THE SHAPE EDITOR

The following presentation shows a step by step procedure of how to create a part program using the Anilam 5000 Control.

This procedure details the use of the interactive CAM that creates the part profile through simple entry of geometrical elements (points, lines and circles) without having to calculate intersections, tangency points etc. These elements are then connected or "chained" to complete the shape.

Creating the Part Program File in the PROGRAM page.

(1) From the MANUAL mode press: Program

NOTE: There are 2 methods of creating a program file, both are shown below.

Method No.1 Creating a new program file.

(2) Press: **Create**   At the prompt NEW PROGRAM: _ type in the program name

NEW PROGRAM: EXAMPLE_ and press

NOTE: Program names can be up to 8 characters in length, but may not include spaces or periods.

Method No.2 Copying and using an existing program.
If a similar program to that required already exists, it may be copied and given a new name. This allows similarly formatted programs to be used without having to re-type the information.
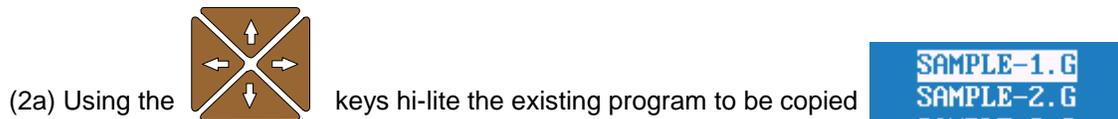
(2a) Using the   keys hi-lite the existing program to be copied SAMPLE-1.G SAMPLE-2.G

Press: **Utility**   this brings up a menu, Copy Rename   with Copy hi-lited

press   This will display a second menu Other... Hi-lite Other and press

This will bring up a third menu Copy to: EXAMPLE_ type in the new program name
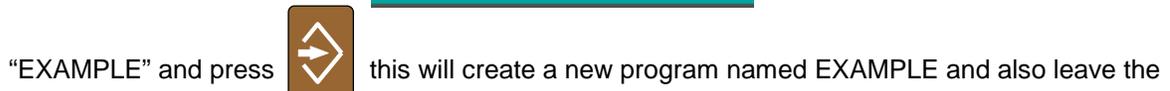
"EXAMPLE" and press   this will create a new program named EXAMPLE and also leave the

original program "SAMPLE1.G" intact. This copied program can now be edited to suit the new part. It is not necessary to type the file extension (.G) as this is completed automatically.

- High light the name the needs the program for, press **CAM**
- The machine program can be coompletely produced in CAM.

Line tool
Arc tool
Geometry tool
Corner radius
Corner chamfer
Chaining tool
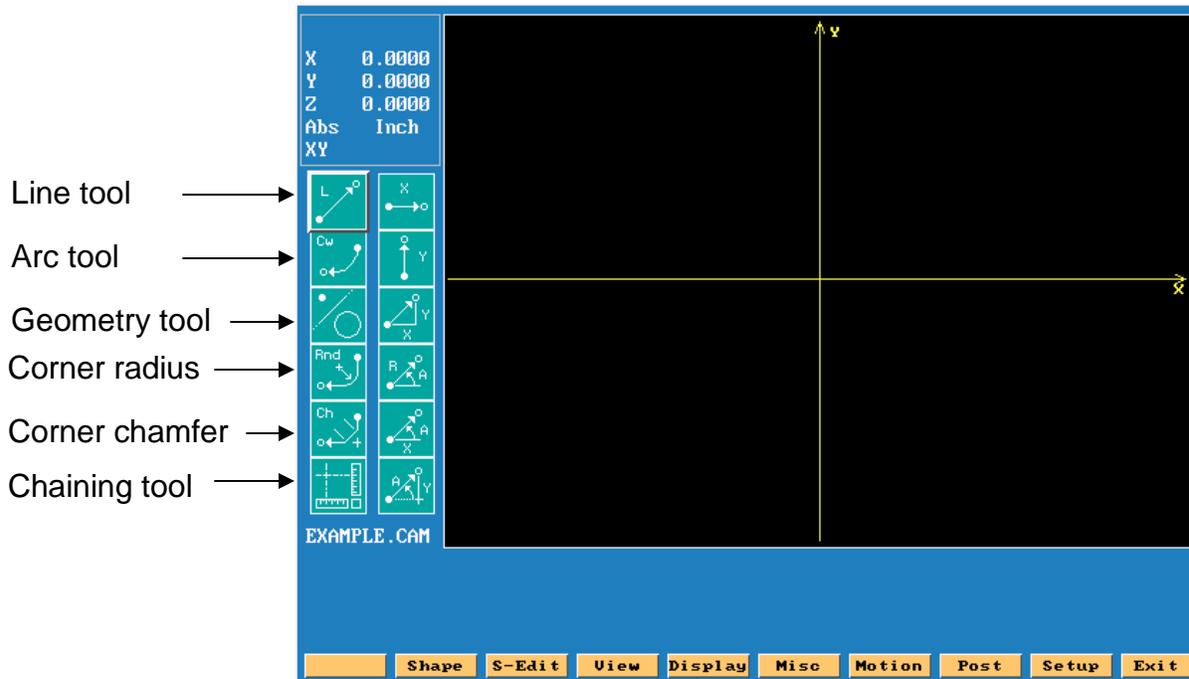
When **Shape** is press, soft keys will change as shown below.

| F1 | F2 | F3 | F4 | F5 | F6 | F7 | F8 | F9 | F10 |
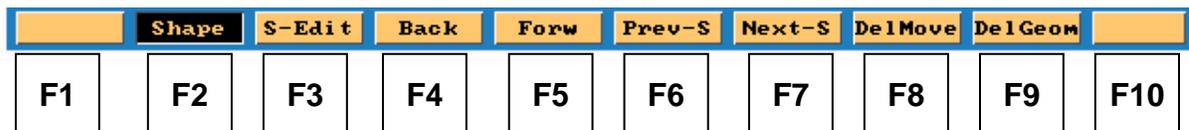
F4- **Back**   Moves curser backwards on a shape.
F5- **Forw**   Moves curser forward on a shape.
F6- **Prev-S**   Moves curser to previous shape.
F7- **Next-S**   Moves curser to next shape.
F8- **DelMove** Deletes last move in shape.
F9- **DelGeom** Deletes geometry, an element number is required.

Press **Shape**   again to turn OFF and return previous funtion keys

Pressing **S-Edit** the following pop-up menu will appear
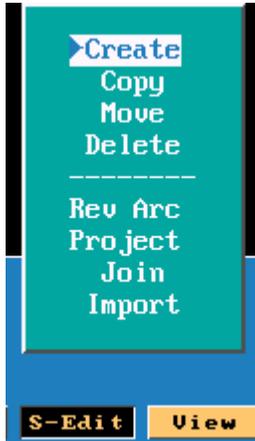
Create   Create a start point for a shape**.**
Copy   Copy a shape to another location.
Move   Move a shape to anew location.
Delete   Delete a shape.
Rev Arc   Reverses direction of an arc in a shape.
Project   Replaces a radius and joins lines.
Join   Connects lines together.
Import   Imports shape from another CAM file.

Pressing **View** the following pop-up menu will appear

XY plane
XZ plane
YZ plane
Isometric

Pressing **Display** the following pop-up menu will appear

▶Fit
Window
Redraw
Half
Double
Scale
Pan
Erase
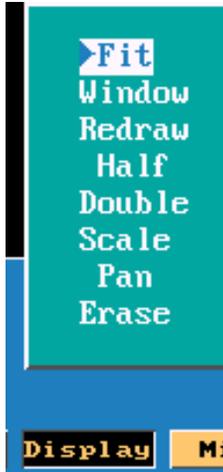
Fit to screen.
Zoom in on windowed area.
Redraw screen
Half size of screen.
Double size of screen.
Scale screen.
Pan move part around on screen.
Erase screen.

Display   Mi

Pressing **Misc** the following pop-up menu will appear

▶Shapes List
Geometry List
Recover Shapes
Recover Paths
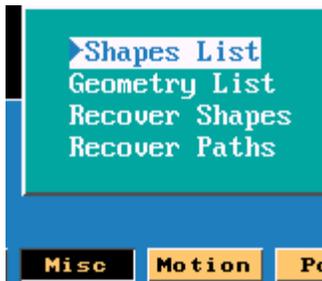
List shapes by number.
Lists all geometry. Showing coordinates.
Recover a deleted shape.
Recover a deleted tool path.

Misc   Motion   Po

Pressing **Motion** the following pop-up menu will appear

▶Contour
Pocket
Drill Pattern
Edit
Delete
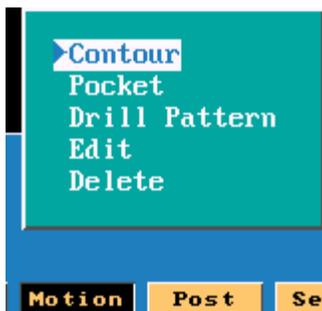
Generate a tool path around a shape.
Generate tool path for pocketing a part.
Drilling cycles and paths.
Edit any of the above paths.
Delete a tool path.

Motion   Post   Se

Pressing **Setup** the following pop-up menu will appear

```
▶Settings..▶
 Shapes....▶
 Geometry..▶
 Paths.....▶
 Post......▶
```

.sc Motion Post Setup
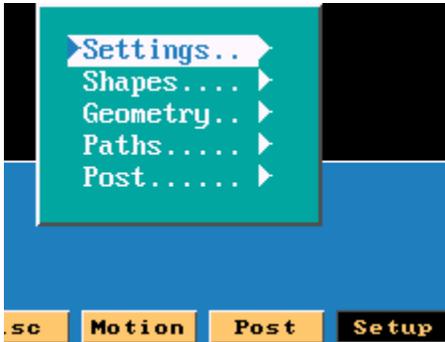
Parameters for CAM.
Turn shapes ON/OFF.
Turn geometry ON /OFF.
Turn tool paths   ON/OFF.
Parameters for post.

When high light is on setting press ⟐ the pop-up menu will appear.

```
▶Dimensions...      Abs
 Input Units..     Inch
 Arrows.......       On
▶S Labels.......     On
 S Axes.........     On
 G Grid.........   None
 P Grid size....   1.00
 P
```
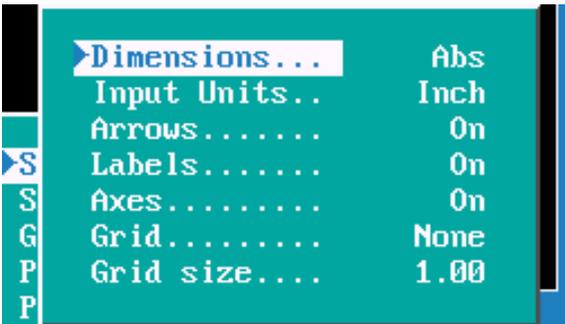
Type of dimensioning ABS/INC.
Units INCH/MM.
Turns Arrows ON/OFF.
Turns ON/OFF element labels.
Turns Axis marrkers ON/OFF.
Turns grid ON/OFF.
Size of grid.

When high light is on paths press ⟐ the pop-up menu will appear.

```
          G—Code Configuration

▶G-Code File Name..    EXAMPLE.G
 Overwrite File....         Yes
 Dimensions........         Abs
 Output Units......        Inch
 Axes..............    Suppress
 Text..............          On
 Program Number....         100  ..
 Block Numbers.....           ▶  ..
 Tool Change.......           ▶  ..
 Format............           ▶  ..
                                 ..
```
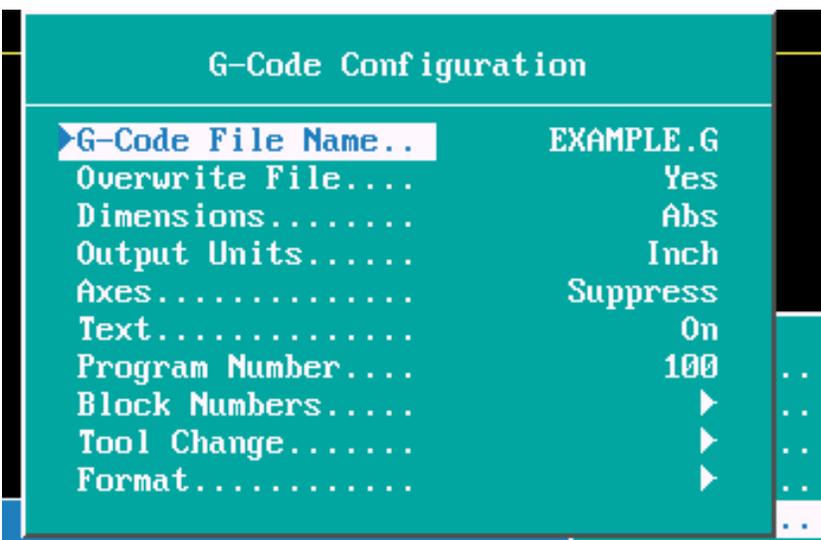
Out-put program name.
Over write existing program.
Out-put type ABS/INC.
Unit INCH/MM.
Only out-put Axis if it moves.
Text on while posting.
Program number.
Block number.
Tool change requirements.
Format Number of decimals

5.

The part below needs to be pocketed.

6.25"

2" rad

3.75"

X0 Y0

4"rad

1.8"rad

6.
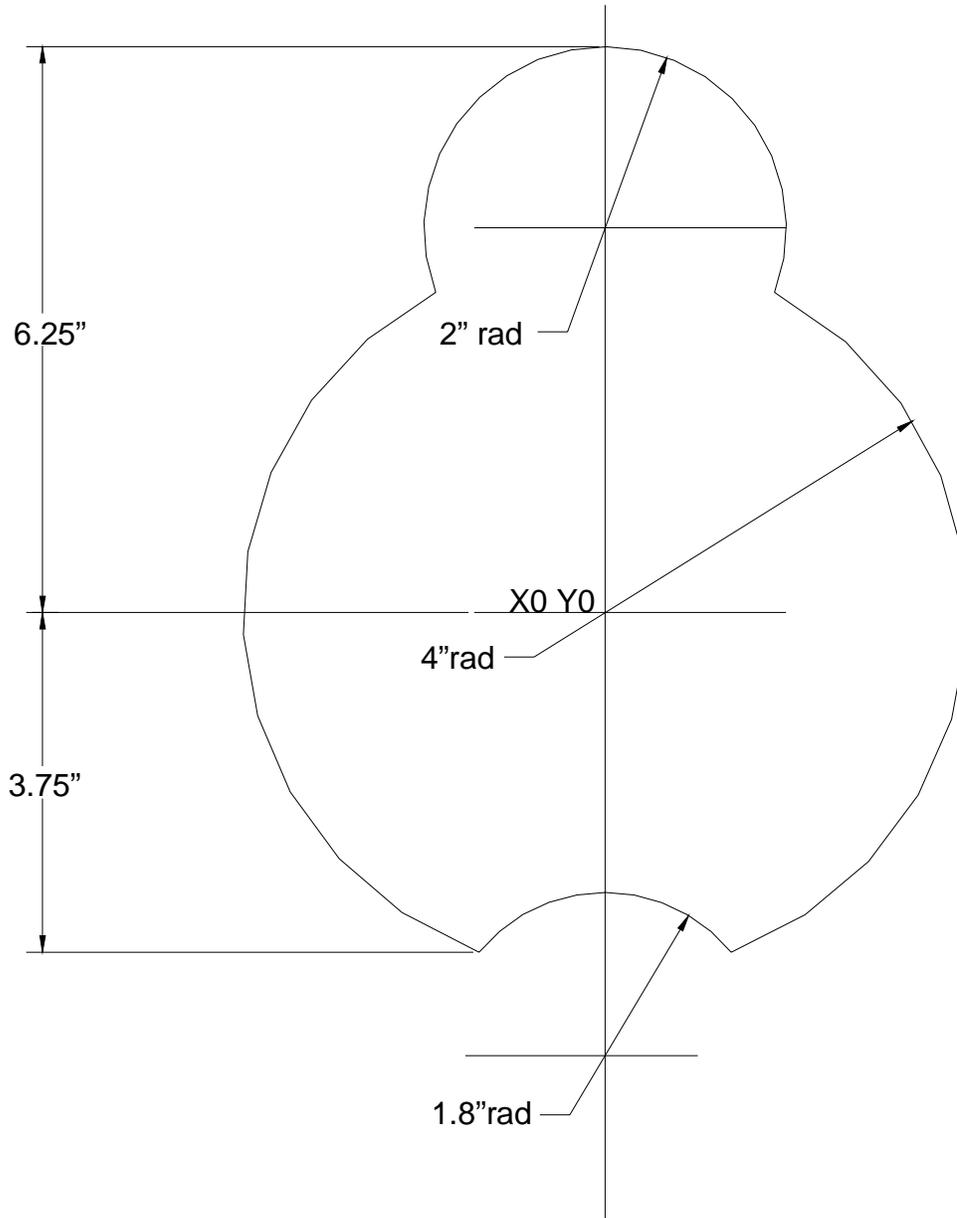
When **CAM** is accessed The high light will be on the top left icon,
for the following exercise the third icon down is the one required.

EXAMPLE.CAM

Press down arrow key twice icon on right will become point definitions.

Enter a point using X & Y coordinates.

Increamentally move from an existing point.

Move an existing point using an angle and a radius.

Center of a circle.

Intersect between two elements.

Using an existing point.

7.
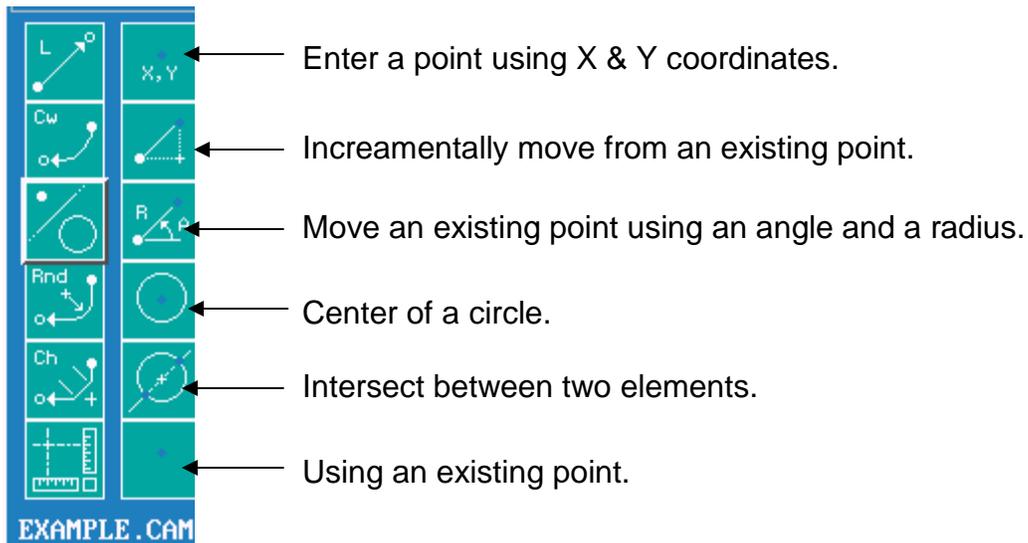
Press [icon] icon on right will change to line definitions.

A line along X axis.

Aline along Y axis.

Aline between two points

A line through a point at an angle.

A line parallel to an existing line.

A line tangent to a circle through apoint.

EXAMPLE.CAM

Press [icon] to get to circle icons.

Circle between two existing elements.

Circle knowing center point and radius.

Circle tangent to a line through a point knowing radius.

Circle tangent to a line knowing center point.

Line between two circles, four options.

Line tangent to a circle through a point.

EXAMPLE.CAM

High light third Icon down on left.

Press [icon] right and then up.
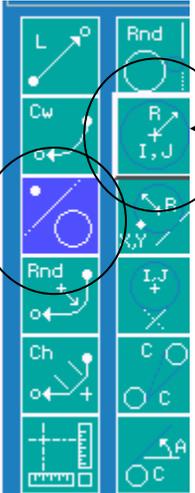
Circle icon high lighted

Geometery icon
turn blue.
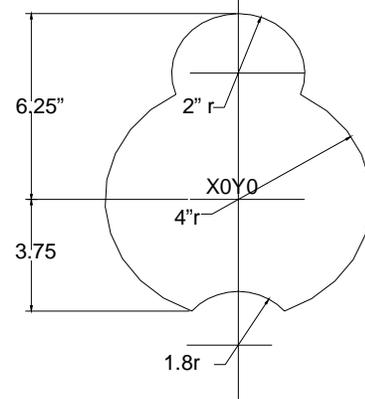
EXAMPLE.CAM

Press [key]   Enter R value: _   R value =4

Press [key]   EXAMPLE.CAM
                Select center definition ..
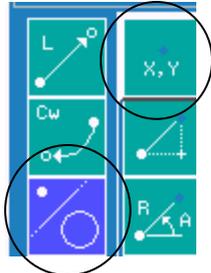
It will now ask for center definition and point definition will appear in right column
of icons. The top icon will be high lighted.

Press [key] it will ask for an value in X0 and Y0 [key] twice.

When entry is 0 (zero) ,it is not required to press the 0 key.

6.25"   2" r
          X0Y0
          4"r
3.75

1.8r

Press [F5] [Display] Fit and screen show the circle that was just entered.

6.25"

2" r

X0Y0

4"r

3.75"

1.8"r
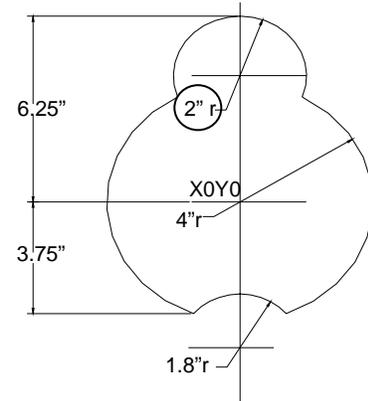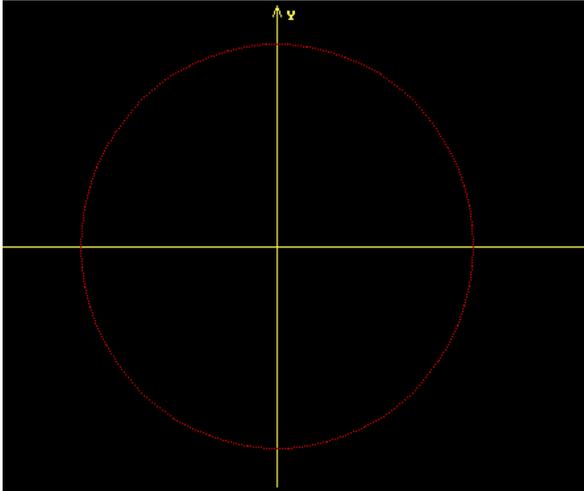
Using same icon put in 2" radius ccircle.

Press enter 2" for radius

Press press to select point value X0 , value Y 4.25 (6.25-2) press

Two circle will now show up on screen.

2" Radius

4" Radius

10.
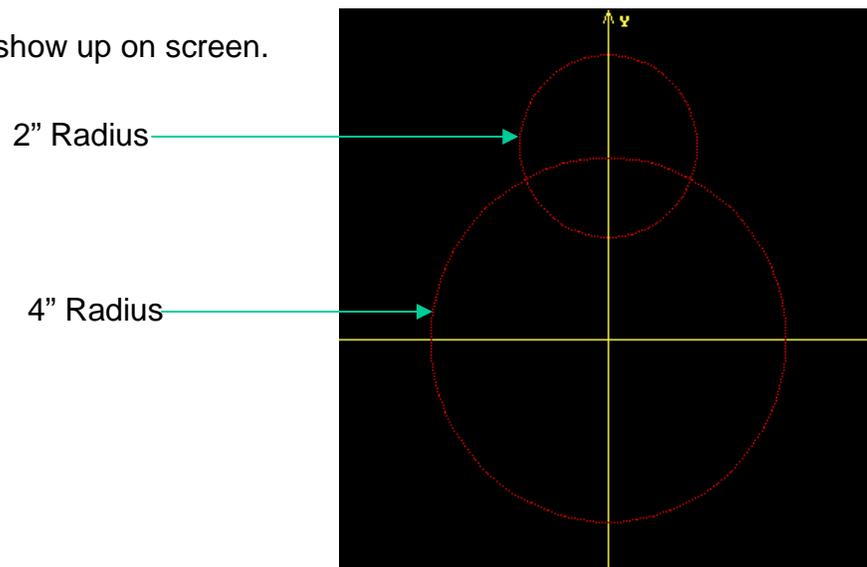
The next element required is the 1.8 radius arc.

In order to do this it is necessary put in some constrution geometry.
First a line has to be draw at -3.75 in the Y axis.

First high light geometry icon as shown below,

press until line definition appear.

Use arrow keys to get to circled line definition.
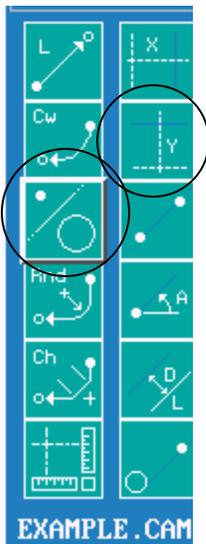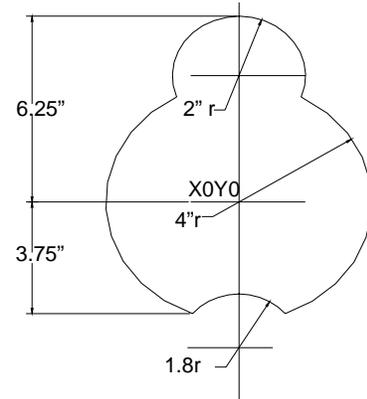
Press enter a value of -3.75
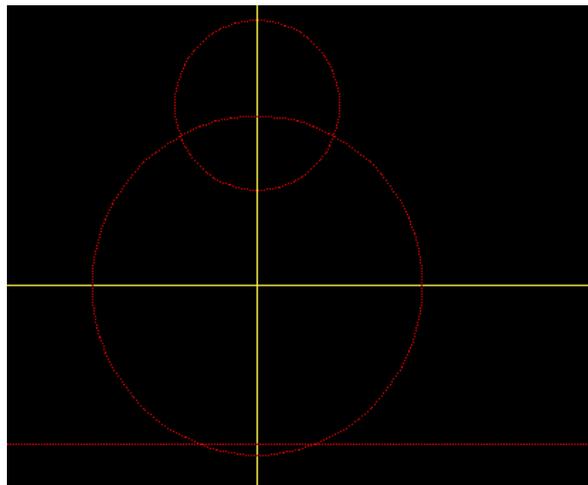
The next geometry required are two points at the intersect of the 4" radius circle and the -3.75 line. To do this use icon circled below.
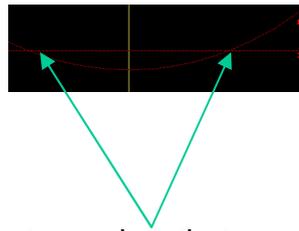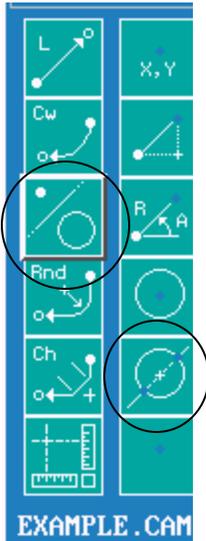
These are the two points that needed to be found.
They are the intersect of circle #1 and line #3.

Press Enter number of first element 1 press second element 3.
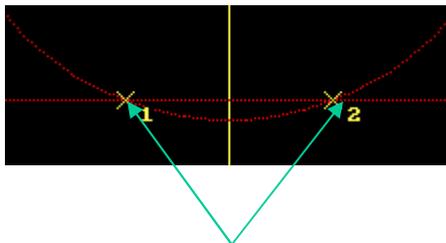
There are now two selections, press 1 press
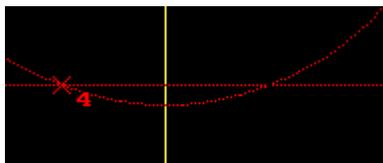
Thre is now an element #4 on left intersect.
Do the same again only select #2.

12.
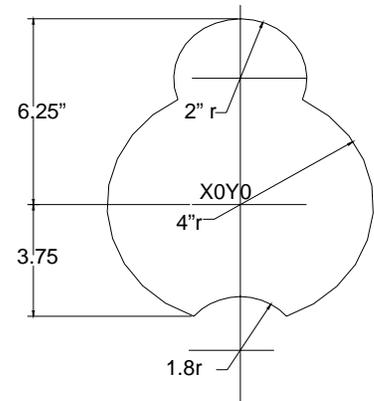
It is now posible to drive an arc between points #4 & #5.
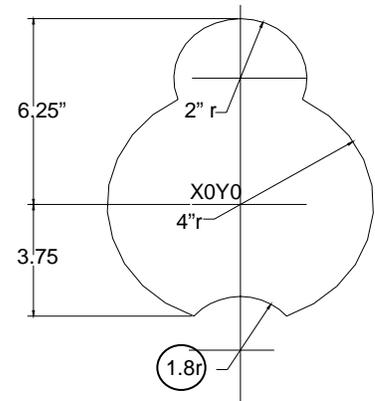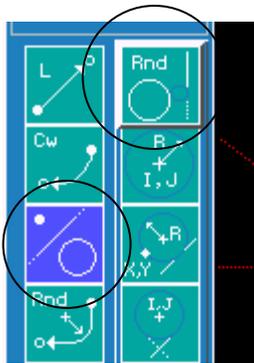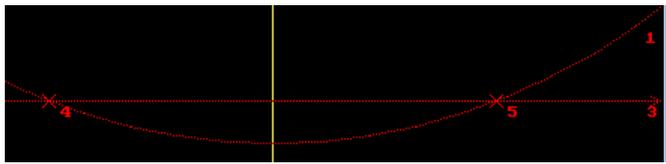
Use the Icon that allows a circle between two elements.

First question R value.   `Enter R value: 1.8_`   Press

Second question   `Enter number of first element: 4_`   Press

Third question   `Enter number of second element: 5_`   Press
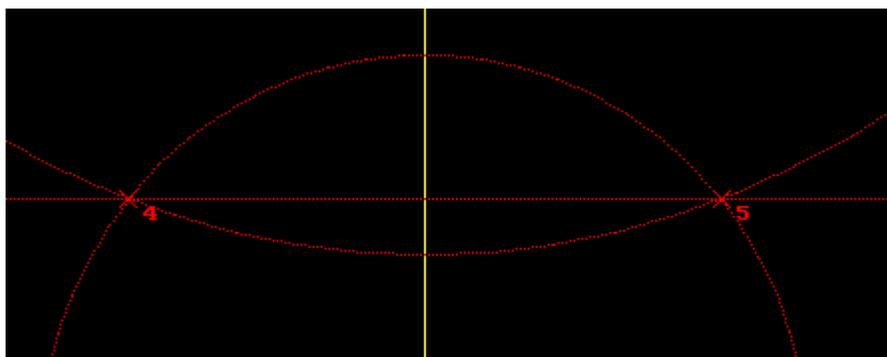
It is necessary to figure the best place to start.
On this part the top will allow us to completely
clean this shape.

Point required for start

6.25"

2" r

X0Y0

4"r

3.75

1.8r
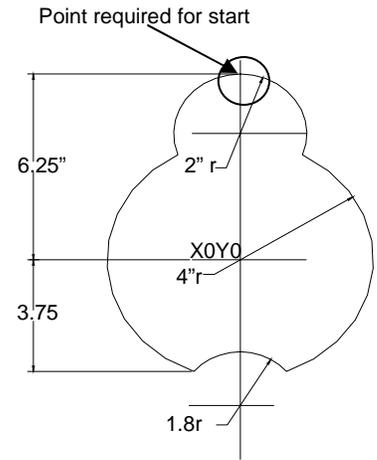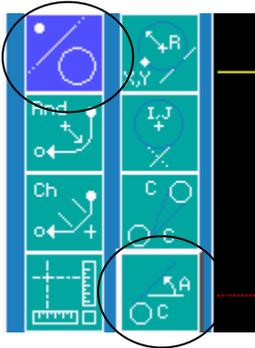
Use icon circled line tangent to a circle at an angle.

First question   Enter angle: 0_   Press

Second question   Enter number of circle: 2_   Press
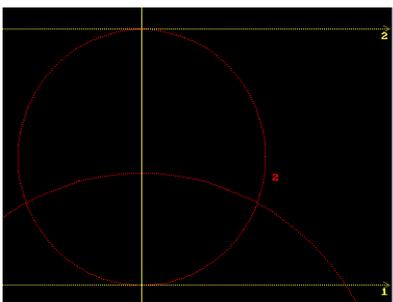
There are two chooses #1 or #2 we will
take #2

![ANILAM]
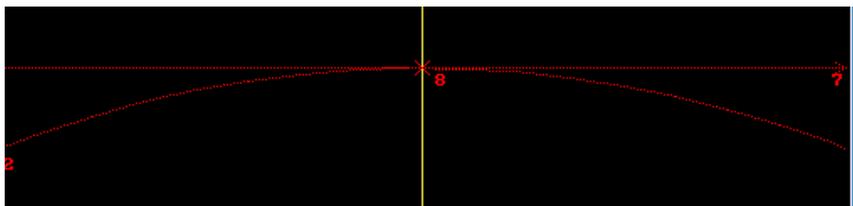
All the necessary geometry has been established, the next thing is to make a shape, to pocket and contour.

Press **F3**   **S-Edit**

>Create
Copy
Move
Delete

Rev Arc
Project
Join
Import

A start point needs to be established for are shape, high light will be on **Create.** Create will set where the shape is going to start.

Press ⟐ select point definition, Use the icon circled.

EXAMPLE.CAM
(1) Select point definition

15.
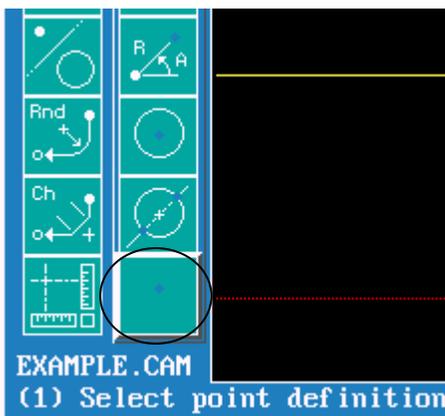
Press ⬦ `From point: 8_` Enter 8 when it says "From point"
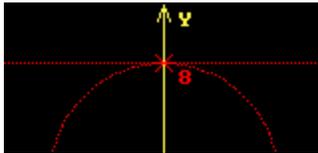
Press ⬦

The white number one is the start point for shape #1.

The icon circle is the construction icon,

high light it press ⬦ it will ask for

an element tobe selected.
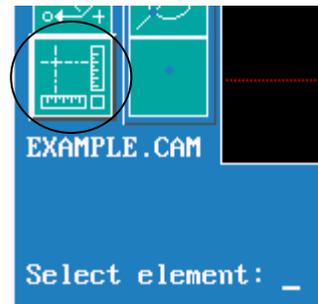
`Select element: -2_` Enter -2 press ⬦

When selecting circles if cutting in clockwise the number is positive if counter clockwise the number is negative as shown above.
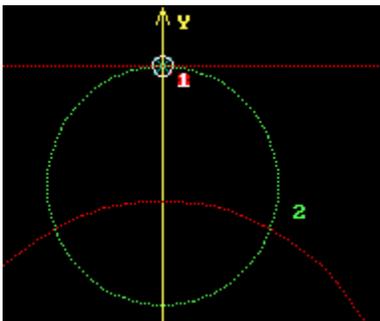

Selected element will show up in green.

The next selection is -1 and there are selection 1 and 2, inthis case 2 is required.

Press 2 ➡
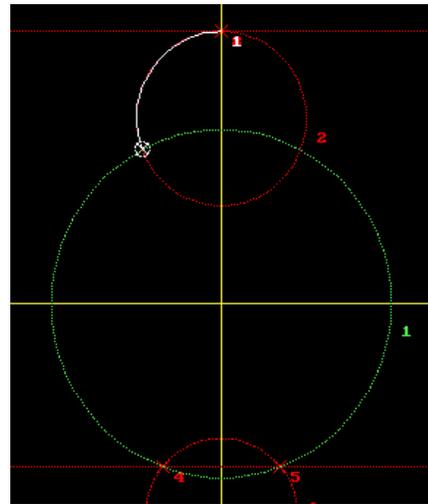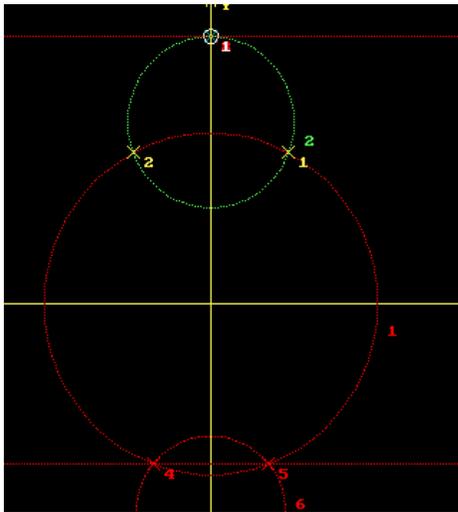
Notice that after 2 was selected element #2 when from green to white and element #1 became green.

The next element is #6, this is positive because the direction is clockwise.

Press 6 ➡ select #2 ➡

press -1 ➡ select #2 ➡

press-2 ➡ select#2 ➡

press 8 ➡
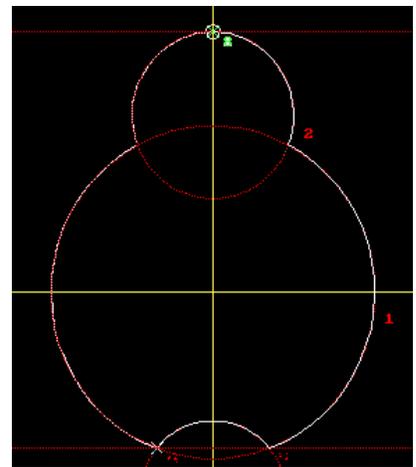
Notice the shape is now outlined in white.

The shape is now complete press | **F9** | **Cancel**

The geometry and shape are complete, the next is to produce a tool path to pocket and contour.

Press   F7   **Motion**

>Contour
 Pocket
 Drill Pattern
 Edit
 Delete

The first tool path is **pocket** using arrow keys high light pocket press.

 Contour
>Pocket
 Drill Pattern
 Edit
 Delete

Press

```
        Pocket Parameters

  Shape number......        1
 ▶Tool diameter.....    0.5000
  XY stepover.......    0.3500
  XY stock..........    0.0100
  Z step............    0.1000
  Z stock...........    0.0050
  Approach height...    0.1000
  Top of pocket.....    0.0000
  Bottom of pocket..   -0.5000
  More..............        ▶
```

```
▶Direct
 Tool table
```

```
 Pocket Parameters

▶Direct..
```

```
  1.▶      0.5000
  2.       0.1250
  3.       0.0000
  4.       0.1250
  5.       0.5000
  6.       0.1560
  7.       0.1870
  8.       0.7500
  9.       0.5000
 10.       0.4500
 11.       0.0000
 12.       0.0000
 13.       0.0000
```

Tool table

There are two methods of entering tool diameters, direct just type in value and select it from toll table.

A pop-up window will appear as above.

To enter a value in any of these parameters first press

enter value press

When **More** is reached it will bring up anew pop-up window.

Second page of parameters.

```
              Pocket Parameters

     Comment..............             ▶
     Interference check..             On
     Angle of cut........           200.0
     Direction of cut....           Forw
     Start point.........          Default
     Tool path color.....          [green]
     Shape Reversed......             No
     Entry Move..........             ▶
     Exit Move...........             ▶
   ▶Machine setup.......              ▶
```
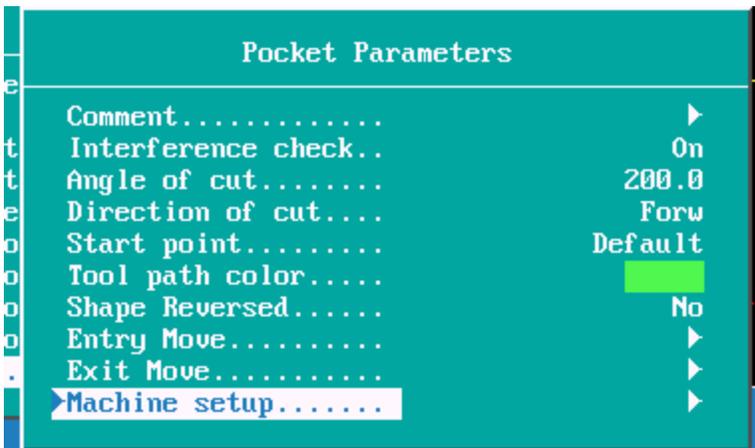
Angle of cut can be left at default or an angle entered, on this part an angle of 200 deg's will be entered.

```
▶Default
 Angle
```

Entry and Exit Move
```
▶None
 Linear
 Circular
```
these are the three options.

Linear is straight line move on to start point, circular ram on move.

```
            Entry Move Setup

    ▶Move Type.....            None
     Arc Length....          0.0000
     Arc Radius....          0.0000
     Origin Point.. {  0.0000,  0.0000}
```
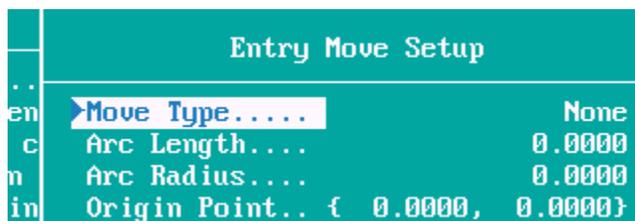
Arc Length is the angle arc ramping on move.
Arc Radius size ram radius.

When **Machine Set** is hight lighted will go to a new pop-up window.

```
          Machine Setup                    han    Tool Change Configuration
ent
rfe   Tool Change.......        ▶       t a    Output Tool Change.. Yes
e o   Initial move......       2D       t a   ▶Tool number.........   1
cti   Coolant at start..       On       te
t p   Coolant at end....     None
 pa   Feedrate..........     20.0
e R   Z Feedrate........      5.0
y M   Spindle at start..     Forw
 Mo   Spindle at end....     None
ine  ▶Spindle speed.....    2,000
```
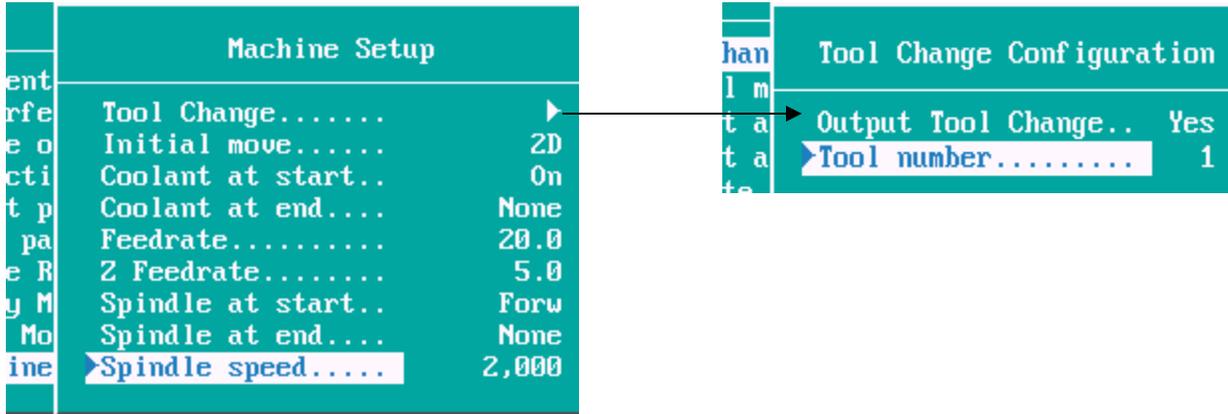
Coolant turn ON at start but not OFF at end as the same tool will be used
for pocketing and contouring. The same applies with spindle.

Press **F10**   **Cont**   Press **F10**   **Cont** Press **F8**   **Calc**

On the right is how it will appear on screen.
Red is geometry.
White is the shape outline.
Green is tool path.

```
Save toolpath?
 Yes      No
```

F1   **Yes**

21.
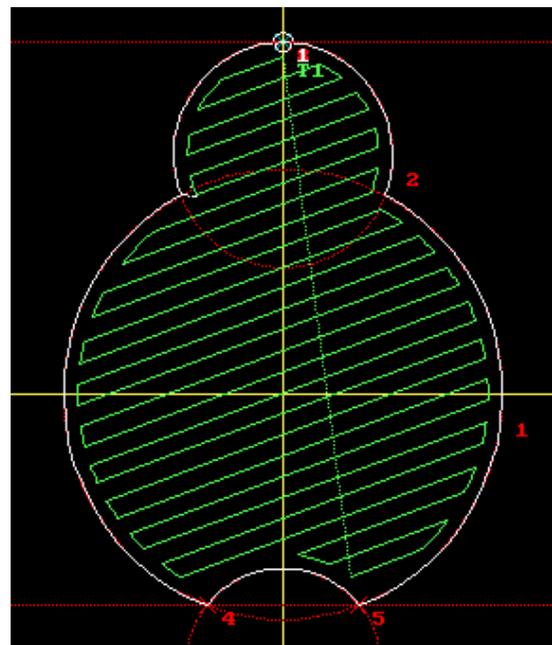
The Contour now need to be done as the edges are still rough.

Press | **F7** | **Motion**

>Contour
Pocket
Drill Pattern
Edit
Delete

press

Contour Parameters

Shape number......... 1
Tool compensation.... CAM Left
Tool diameter........ 0.5000
XY stepover.......... 0.0100
Number of XY passes.. 3
Z step............... 0.0000
Approach height...... 0.1000
Top of contour....... 0.0000
Bottom of contour.... -0.5000
Stepover direction... Toward
>More................ ▶
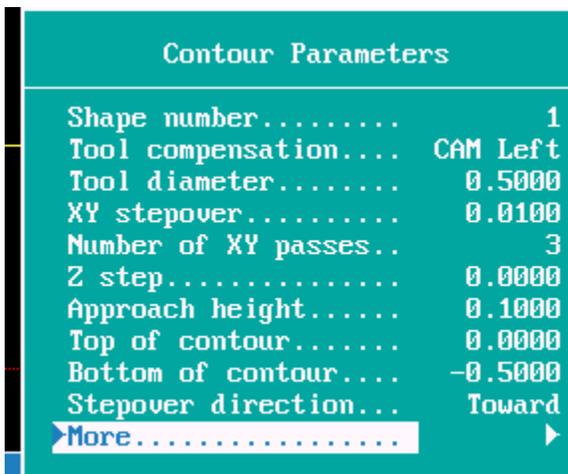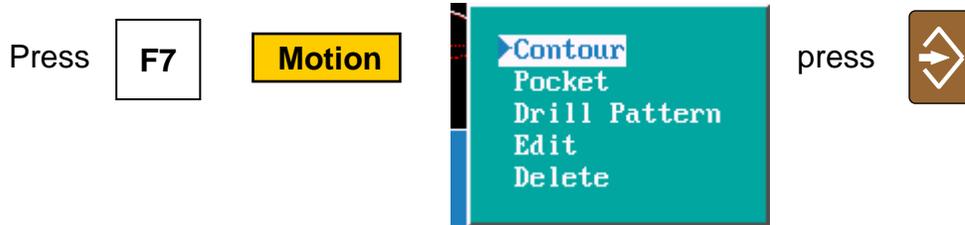
It now needs to know which side to put tool comp.

Stepover this is around contour only.
Number of passes around contour.

Contour

Contour Parameters

Shape number.
Tool compensa
Tool diameter
XY stepover..
Number of XY
Z step......
Approach heig
Top of contou
Bottom of con
Stepover direction... Toward
>More...............

Comment............. ▶
Interference check.. On
Tool path color.....
Shape Reversed...... No
Entry Move.......... ▶
Exit Move........... ▶
>Machine setup....... ▶

Entry Move Setup

Co
In >Move Type..... Circular
To Arc Length.... 90.0000
Sh Arc Radius.... 1.0000
En Origin Point.. { 0.0000, 0.0000}
Ex

Do this for both Entry and exit moves.

There is no need for a tool change because the same tool is being use for the contour as pocketing.
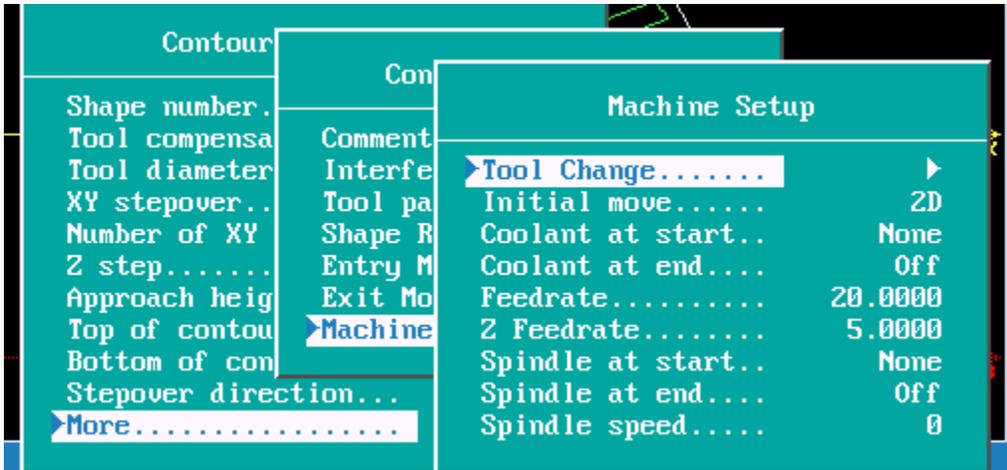
Note: Coolant and spindle at start are entered as **None** and turned **Off** at end, as they are still on from pocketing.

Pocket tool path #1

Profile tool path #2
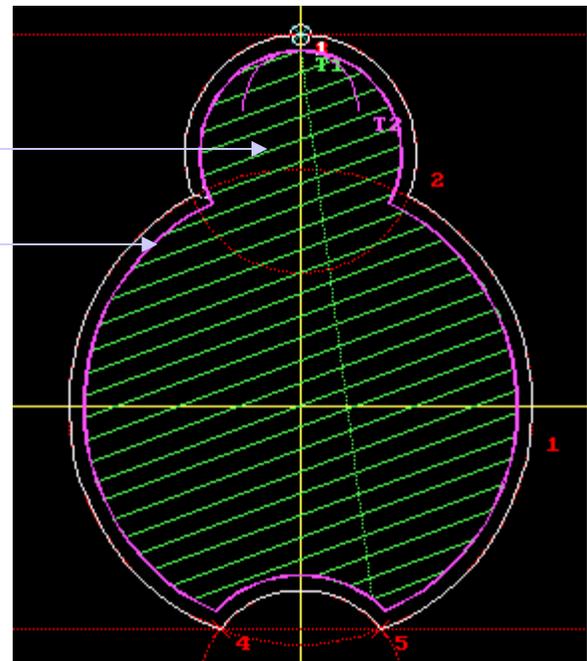
| F9 | Setup | Arrow down to post |

Settings.. ▶
Shapes.... ▶
Geometry.. ▶
Paths..... ▶
▶Post......

G-Code Configuration

▶G-Code File Name..   EXAMPLE.G
Overwrite File....   Yes
Dimensions........   Abs
Output Units......   Inch
Axes.............   Suppress
Text.............   On
Program Number....   100   .. ▶
Block Numbers.....   ▶   .. ▶
Tool Change.......   ▶   .. ▶
Format...........   ▶   .. ▶

Tool Change Configuration

▶XY Location.........{ 0.0000,   0.0000}
Z Location........   0.0000
Tool Change Format..   T%02DM06
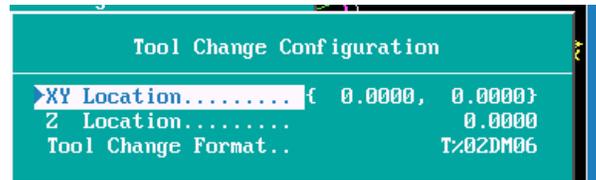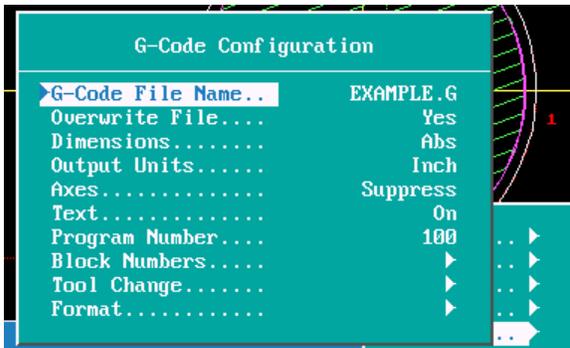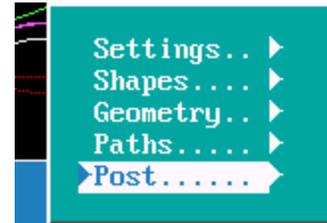
If machine has a tool changer arrow down to Tool Change press enter arrow down to Tool Change Format if M06 is required it should read as follows **T%02DM06**.

Block Number are set to start at 10 and increament by 10 this can be change dependinng on your preferance.
Format is set for 8 decimal place change these to 4.

Press | F10 | Cont | press | F9 | Setup

Press | F8 | Post

When finish press | F10 | Exit | it will now return to Program Page.

High light EXAMPLE.G press | F7 | Draw

press | F5 | Display fit will be high lighted press ⮕
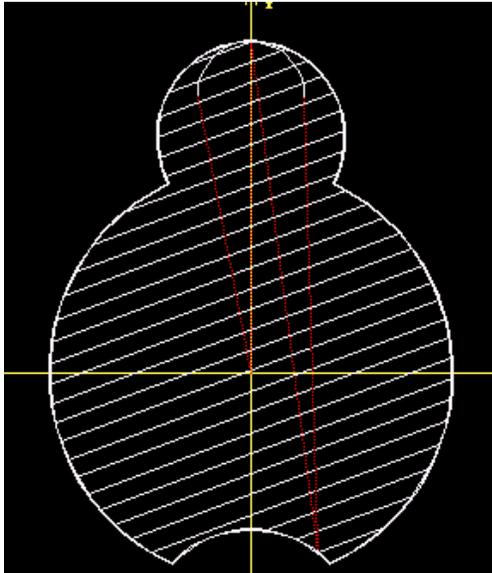
XY view
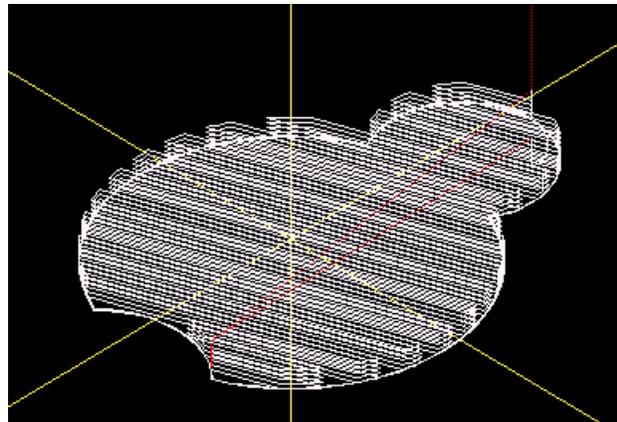

Isometic view

All that is left to do is set tool length offsets and fixture offsets, part is ready to run.

If the G-Code Configuration is now correct it can be save for future use.

Press **↑** **F5** **Display** until top left corner shows

..\ high light EXAMPLE.CAM press **F9** **Utility** high light

will be on **Copy** press **⇨** Arrow down to **Other** Type in

C:\P5M\DEFAULTS.CAM

# 5000M CNC CONTROL
DXF Converter

DXF file can be converted into 5000 machine programs using the Offline software.

The DXF files are stored in the Program Page.

When going to Program Page only .G and/or .M file will be

displayed,press shift **F8** **Display** twice, this will

display all files on **Program** page.

If DXF is on disk it needs to be copied into **C:\User** directory.

press shift **F7** **Log** select **A:**

High light required DXF program press **F9** **Utility** Copy to **C:**

Log back to **C:\User.**

High light DXF program

TOOL2.DXF
F UPARM2.DXF
XE2436-1.DXF
XE2436-1.FXD

Press **F9** **Utility** high light DXF Converter

Copy
Rename
Print
Restore
Communications
Gcode To Conv.
▶DXF Converter
More ...
PR

Press E N T E R

1.

ANILAM

File name
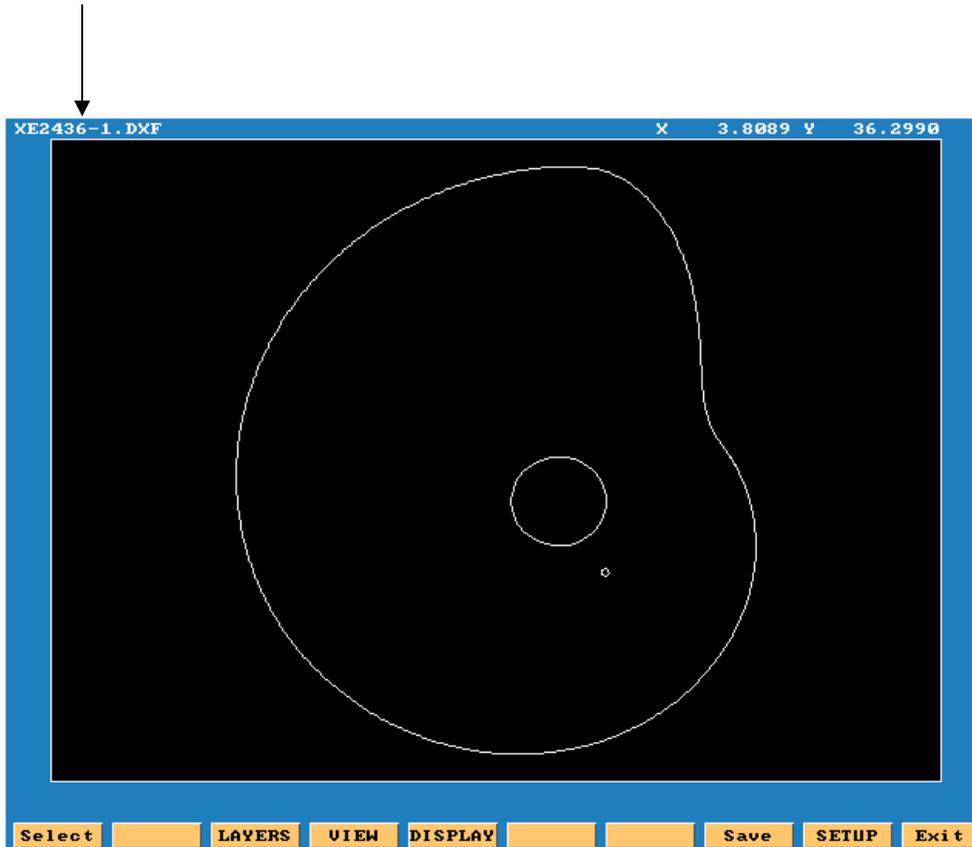
Select: Used when selecting elements on drawing.
Layers: Allows layer on drawing to be turned Off or On.
View: XY,XZ,YZ or isometric.
Display: Fit, window, redraw, half or double.
Save: Saves program with .G once converted.
Setup: Allows setup of inputs and outputs.
**Exit :** Goes back to **Program** page

2.

ANILAM

Zero on most drawings is usually not at a point that is covenient for programing, so there is a way it can be changed.
In the case of current drawing, the center of the hole in center of part is the best point X0 Y0.
To do this Press the Ctlr key and hold it down put mouse point on to circle and press left mouse key, it will change to yellow, release keys.
At the bottom of screen X, Y, and Z cordinate will appear and also circle diameter.

Circle: 16.2194 17.9176 0.0000; 2.1250; 0

Select   LAYERS   VIEW   DISPLAY

Circle center

X   Y   Z
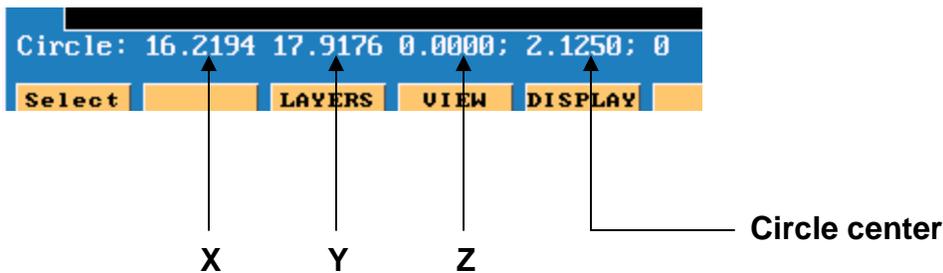
Now press **ALT** key and letter **T** at the same time. This will in put these coordinates in to the **SETUP** and change **X0 Y0** to the center of hole.

Press   F9   SETUP   ▶Output...
                            Display.. ▶   Press   ENTER

Output program name....	XE2436-1	Program name.
Shift X...............	0.00000	Imported X shift.
Shift Y...............	0.00000	Imported Y shift.
Output dimensions......	Absolute	Absolule/increamental.
Create Mode...........	Smart	Smart/Overwrite.
Starting sub number....	1	Starting subroutine number.
Program type...........	G-code/ISO	Output .G and .M format programs.
Output warnings........	Yes	Warns if elements done meet.
Re-calc Intersections..	Yes	Re-calculates intesects.
Output resolution......	5	Numbers behind decimal point..
Create main in new.....	Yes	Create a main program.
Convert values.........	None	Inch / Metric / None.
Convert polyline.......	None	Convert polyline to arc/prompt/none.

3.

Press **F10** Exit

Press **ALT**key and letter **F**key at the same time, this will mark the end of each element.

Press **F5** DISPLAY high light **Window** press ENTER

Fit
►Window
Redraw
Half
Double

A box will appear on screen move around sceen using
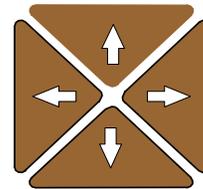
press **F6** Compress position box as shown below press ENTER

4.
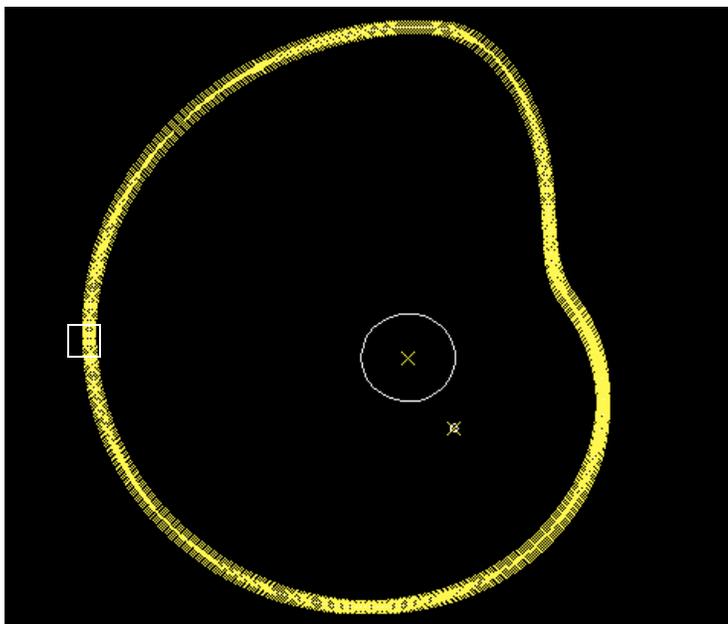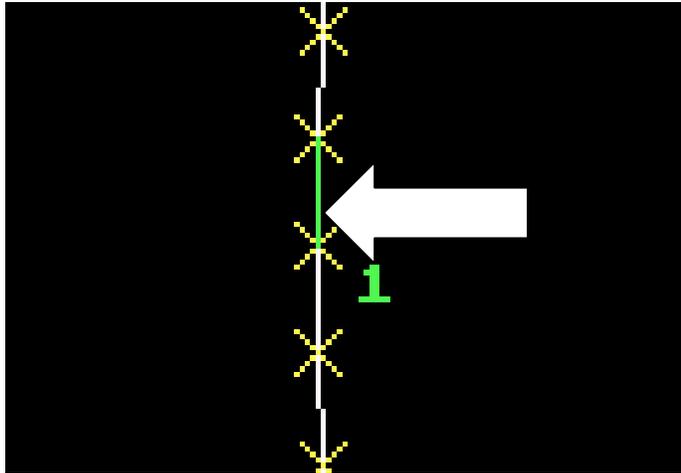
Press **F1** Select point mouse arrow to lower end of a line as shown

and pess left mouse key. Line will turn green as above and put a number at low
end of line, the position of the number is the start point.
Now point to the line below it and press left mouse key, all off the line
will be come green.

Press **ALT** key and letter **F** key at the same time the end of line markers
will disappear.

Press **F5** DISPLAY press **ENTER** Part will appear at full size on screen

5.

ANILAM

Press **F8** [Save]

Succesfully created XE2436-1.M.

| Select | | LAYERS | VIEW | DISPLAY | | | Save | SETUP | Exit |

Press **F10** [Exit]

Exit (Y/N)?

| Yes | No | | | | | | | Cancel | |
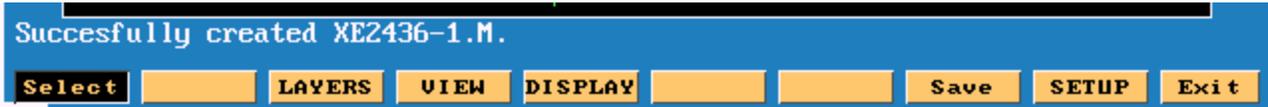
Press **F1** or Press **Y** [ENTER] it will now return to **Program page.**

High light **.G or .M** file press

F4 [Edit]

6.

High light .**M** file press | F8 | [Edit] in coversational will appear as below

```
 1 Call 1
 2 EndMain
 3
 4 Sub 1
 5 Dim Abs
 6 Rapid      X -14.44357 Y 1.05070
 7 Line       X -14.44446 Y 1.17741
 8 Line       X -14.44421 Y 1.30422
 9 Line       X -14.44282 Y 1.43111
```

Start of program

```
719 Line       X -14.40565 Y 0.16721
720 Line       X -14.41445 Y 0.29298
721 Line       X -14.42213 Y 0.41891
722 Line       X -14.42868 Y 0.54500
723 Line       X -14.43410 Y 0.67123
724 Line       X -14.43839 Y 0.79760
725 Line       X -14.44155 Y 0.92409
726 Line       X -14.44357 Y 1.05070
727 EndSub
```

End of program

Program has to be **Edited** , to put in tool changes or cutter comp and **Z** moves.

High light .**G** file press | F8 | [Edit] in G code format will appear as below

```
M98 P1
M2

O1
G90 G0 X -14.44446 Y 1.17741
G1 X -14.44421 Y 1.30422
G1 X -14.44282 Y 1.43111
G1 X -14.44030 Y 1.55807
G1 X -14.43662 Y 1.68510
G1 X -14.43181 Y 1.81218
```

```
G1 X -14.41445 Y 0.29298
G1 X -14.42213 Y 0.41891
G1 X -14.42868 Y 0.54500
G1 X -14.43410 Y 0.67123
G1 X -14.43839 Y 0.79760
G1 X -14.44155 Y 0.92409
G1 X -14.44357 Y 1.05070
G1 X -14.44446 Y 1.17741
M99_
```

Program has to be **Edited** , to put in tool changes or cutter comp and **Z** moves.

7.

This example will show multiple subroutines.
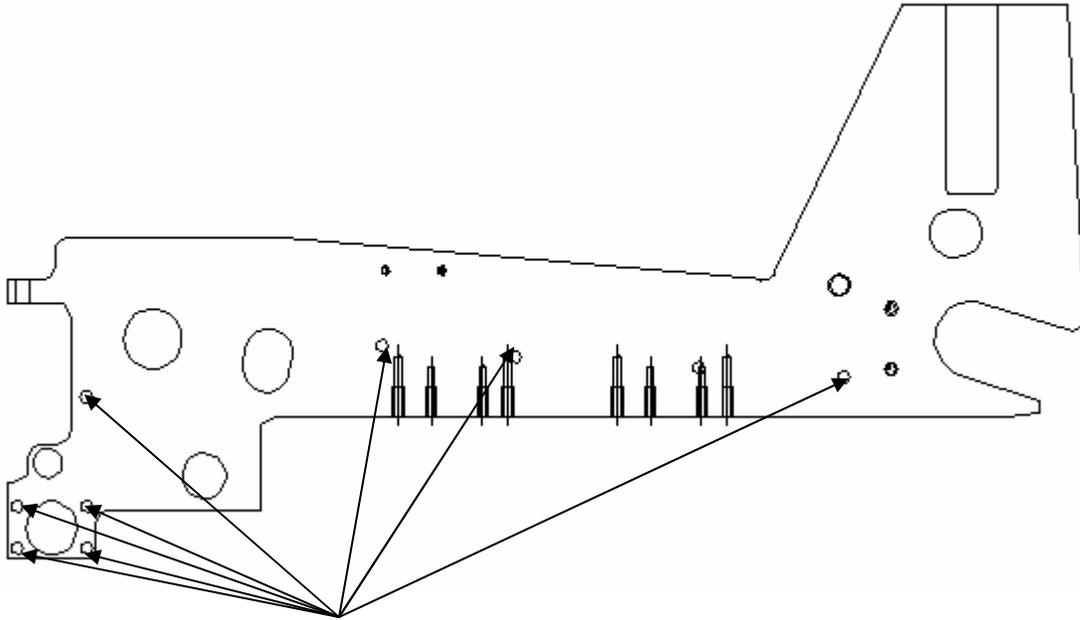
Press **F1** [Select] and pick all holes that are the same size , in this case 8. When going to second set of holes press right key on mouse.

Create new shape (Y/N)?
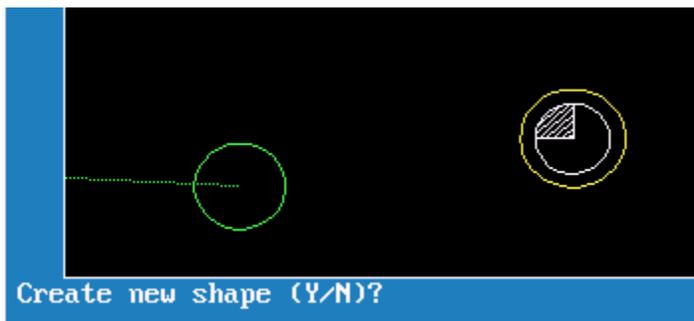
The green circle is the last of previously selected holes. The yellow circle is the one selected with right mouse button and in the bottom left it is asking if this is a new shape, the answer is **Y** .It will put a number 2 next to this hole, meaning this is shape 2.

8.

The print below shows the four shapes of the different size holes.
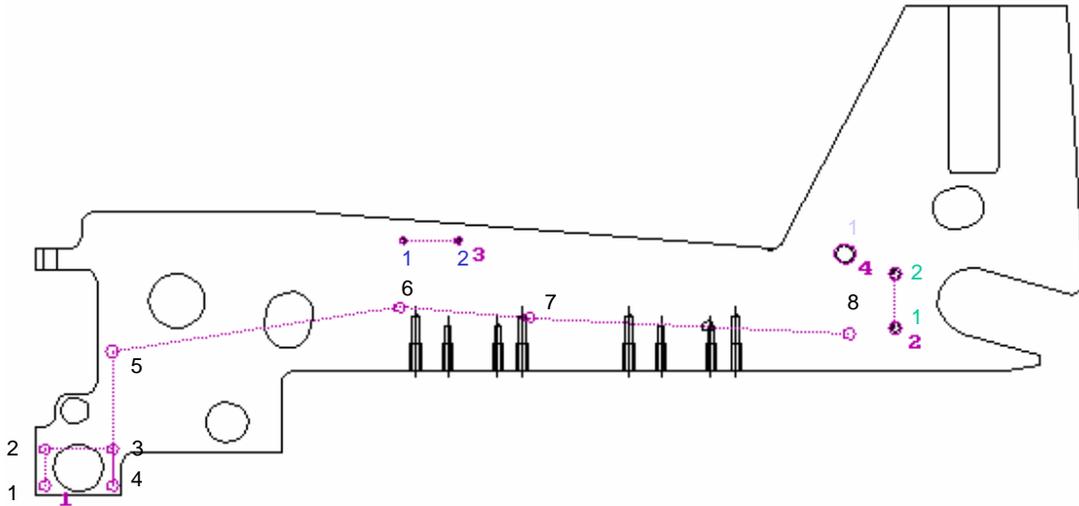
Press **F8** [Save]

Press **F10** [Exit]

9.

Below is the output from the DXF converter.
Some editing is required to put in drilling cycles.

```
M98 P1
M98 P2
M98 P3
M98 P4
M2

O1
G90 G0 X 0.15625 Y 0.84375
G0 X 0.15625 Y 0.15625
G0 X 1.34375 Y 0.15625
G0 X 1.34375 Y 0.84375
G0 X 1.34327 Y 2.66153
G0 X 6.38400 Y 3.50000
G0 X 14.30400 Y 3.00000
M99

O2
G90 G0 X 15.10019 Y 3.12200
G0 X 15.10019 Y 4.12200
M99

O3
G90 G0 X 7.42403 Y 4.75138
G0 X 6.45603 Y 4.75138
M99

O4
G90 G0 X 14.21138 Y 4.51231
M99
```

Subroutine calls

Subroutine for positions of the eight holes numbered in black.

Subroutine for positions of the two holes numbered in green.

Subroutine for positions of the two holes numbered in blue.

Subroutine for positions of the one hole numbered in blue.

10.

The program below is Edited with the drill cycles in the program.

```
N1   G90 G70 G0 T0 Z0
N2   T1
N3   G81 Z-0.1500 R0.1000 F10.0
N4   M98 P1
N5   M98 P2
N6   M98 P3
N7   M98 P4
N8   G80
N9   G0 Z5
N10  T2
N11  G83 Z-0.5000 R0.1000 F12.0 I0.2500
N12  M98 P1
N13  G80
N14  G0 Z5
N15  T3
N16  G87 Z-1.0000 R0.1000 F12.0 I0.2500 J0.0200 K0.1500 W0.0100 U0.5000
N17  M98 P2
N18  G80
N19  G0 Z5
N20  T4
N21  G81 Z-0.2500 R0.1000 F15.0
N22  M98 P3
N23  G80
N23  G80
N24  G0 Z5
N25  T5
N26  G83 Z-1.0000 R0.1000 F8.0 I0.2500
N27  M98 P4
N28  G80
N29  G0 Z5
N30  M2
N31  O1
N32  G90 G0 X 0.15625 Y 0.84375
N33  G0 X 0.15625 Y 0.15625
N34  G0 X 1.34375 Y 0.15625
N35  G0 X 1.34375 Y 0.84375
N36  G0 X 1.34327 Y 2.66153
N37  G0 X 6.38400 Y 3.50000
N38  G0 X 14.30400 Y 3.00000
N39  M99
N40  O2
N41  G90 G0 X 15.10019 Y 3.12200
N42  G0 X 15.10019 Y 4.12200
N43  M99
N44  O3
N45  G90 G0 X 7.42403 Y 4.75138
N46  G0 X 6.45603 Y 4.75138
N47  M99
N48  O4
N49  G90 G0 X 14.21138 Y 4.51231
N50  M99
```

11.

In this example of a full drawing and how to turn off unnecessary information ,such as dimensions etc.
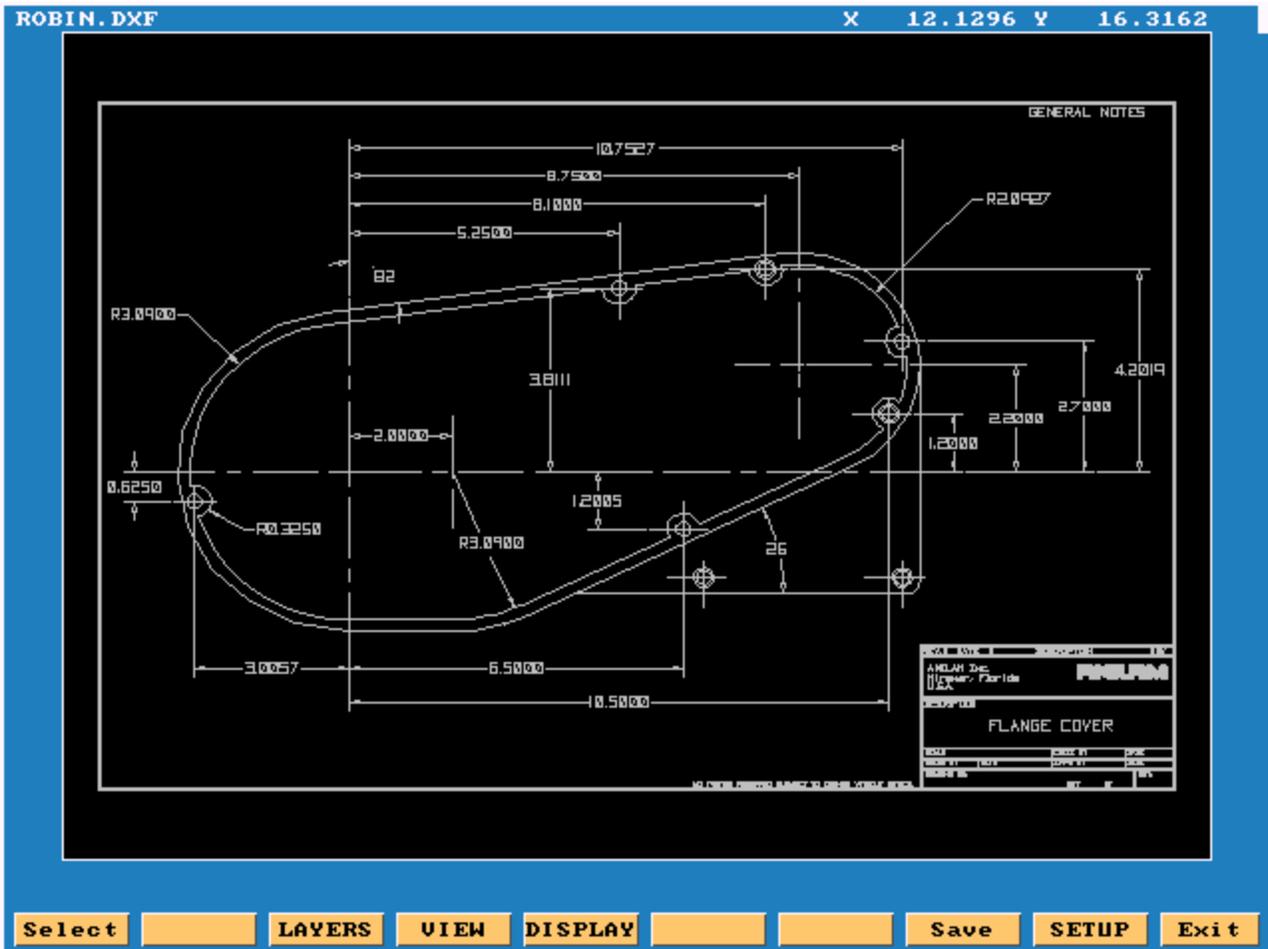
Press F3   Layers

```
All Layers on
Invert Layers
▶Toggle Layers
```

High light **Toggle Layers** press   **E N T E R**

Put high light on layers not required and press   **E N T E R**   to turn OFF.

```
           Layers

  1.▶0.......... Off
  2. AM_TR...... Off
  3. AM_BM...... Off
  4. AM_BL...... Off
  5. AM_VIEWS... Off
  6. AM_PARDIM.. Off
  7. AM_REFDIM.. Off
  8. AM_VIS..... Off
  9. AM_HID..... Off
 10. AM_SUPPR... Off
 11. CONT.......  On
 12. CEN........ Off
 13. HID........ Off
```
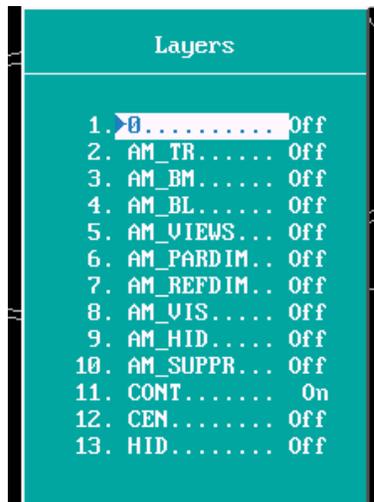
In the drawing shown the only layer required to be left on is #11

13.

Only the part profiles and holes are left.
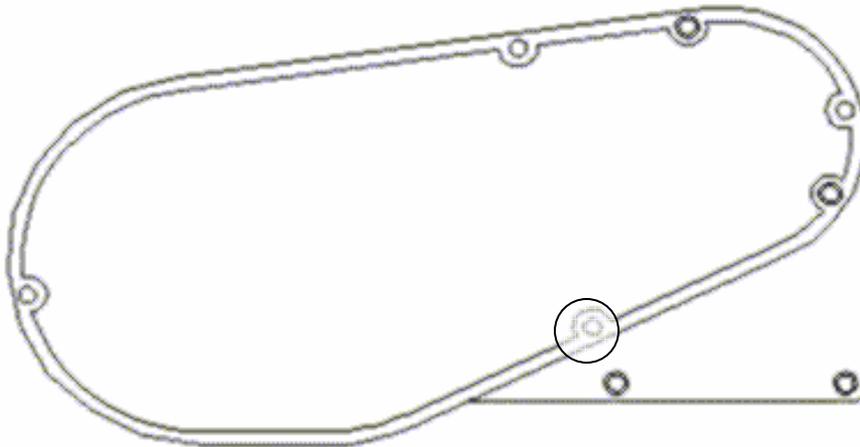
The circled area is blown up below. It show an error in the drawing , where two lines are not connected. It will stop, select next element and the following message will appear.

Entity not connected. Connect anyway (Y/N)?

Press **Y** to continue.
It will then continue around part.

14.